Chapter 1: A Quick Tour of ICS

What is ICS SOFTWARE?

ICS is a uniquely powerful instrumentation control and data analysis software package. ICS is designed to control semiconductor test equipment used for device characterization and other microelectronics testing.

ICS operates in the Microsoft Windows 95, Windows 98, Windows NT 4.0, or Windows 2000 environment. ICS eliminates the need for any programming or front-panel interaction with the instrument. Each instrument supported by ICS is controlled from a user-interface configured by point-and-click editing.

Instruments supported by ICS drivers include:

- Semiconductor Parameter Analyzers
- DC Source Monitor Units
- Curve Tracers
- LCR Meters
- Impedance Analyzers
- Switch Matrices

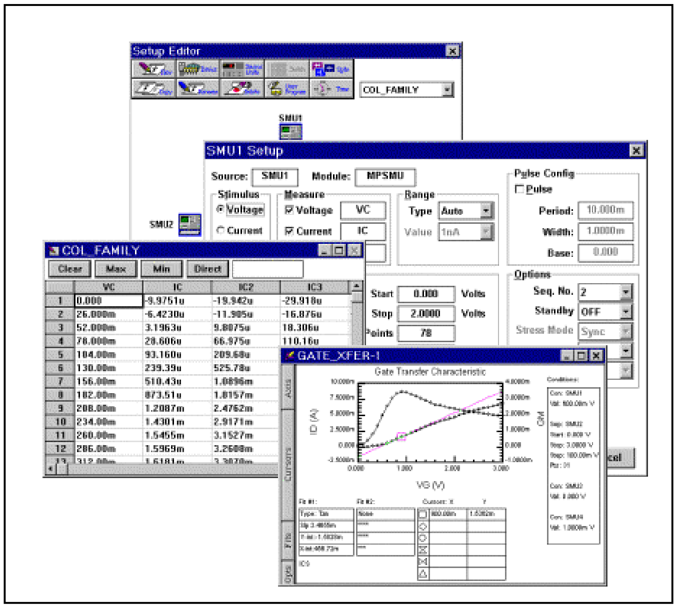


Figure 1-1: ICS

ICS delivers more than programming-free instrumentation control. ICS' spreadsheet windows and scientific plotting capability allow you to view and analyze data easily. ICS also includes valuable scientific and engi-neering data reduction tools necessary for interpreting test results, such as cursor assignments and curve fitting routines. ICS' resources also include a robust file management system. ICS' file management system allows the user to create multiple databases without relying on external database software. Metric's versatility is established by the broad range of driver libraries available for ICS. These libraries provide instrumentation control in both DC parametric and LCRZ applications.

How ICS Stores Information

All of the information that is either specified or measured in ICS 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 add new hardware or change your existing hardware arrangement This means that all you have to do to re-measure a test setup from a cold start of ICS 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 break-down 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. **A Test Setup Defines the Instrument Setup Configuration of Each Source Unit:**

ICS 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 Labelled:**

A source unit conventionally sources voltage and measures current, or sources current and measures voltage. ICS 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, ICS 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 Ids1, Ids2, Ids3, etc. Each data vector corresponds to a single curve. ICS 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 ICS).

4. **A Test Setup Defines the Switching Topology**

When a switching matrix is integrated with the measurement system, the switch-ing topology name is part of the test setup definition. This allows you to create a group of test setups that are distinguished by varying switch configurations.

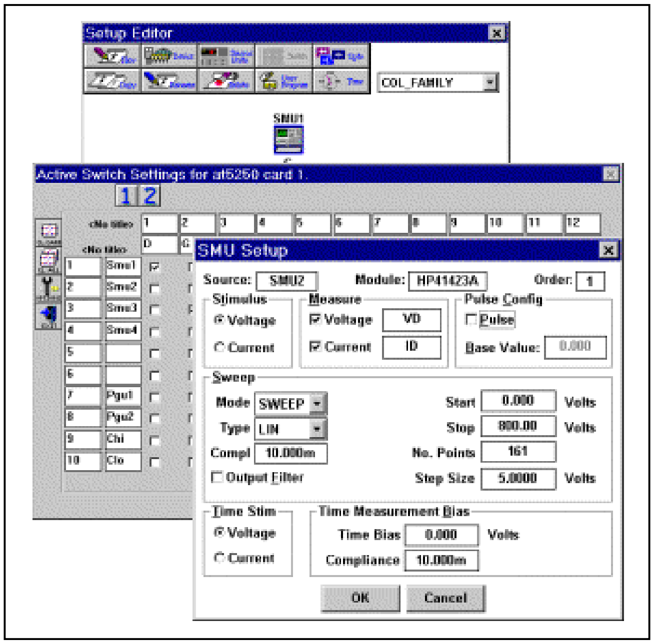


Figure 1-2: The test setup stores instrument setup conditions, documents the instrument/DUT connections, stores what data is returned and and how it is labeled, and stores the switching topology name if a switching matrix is present.

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 ten test set-ups in a single project file in Windows 3.X. 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 Autostore feature, but this is a higher function of ICS covered elsewhere in this manual. In general, the latter point is true.

* A test setup cannot be copied to another project file.

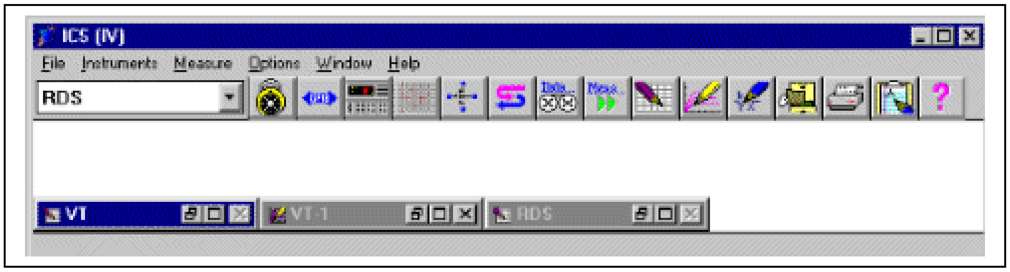


Figure 1-3: Test Setups corresponding to a Project File are displayed along the bottom of the Desktop.

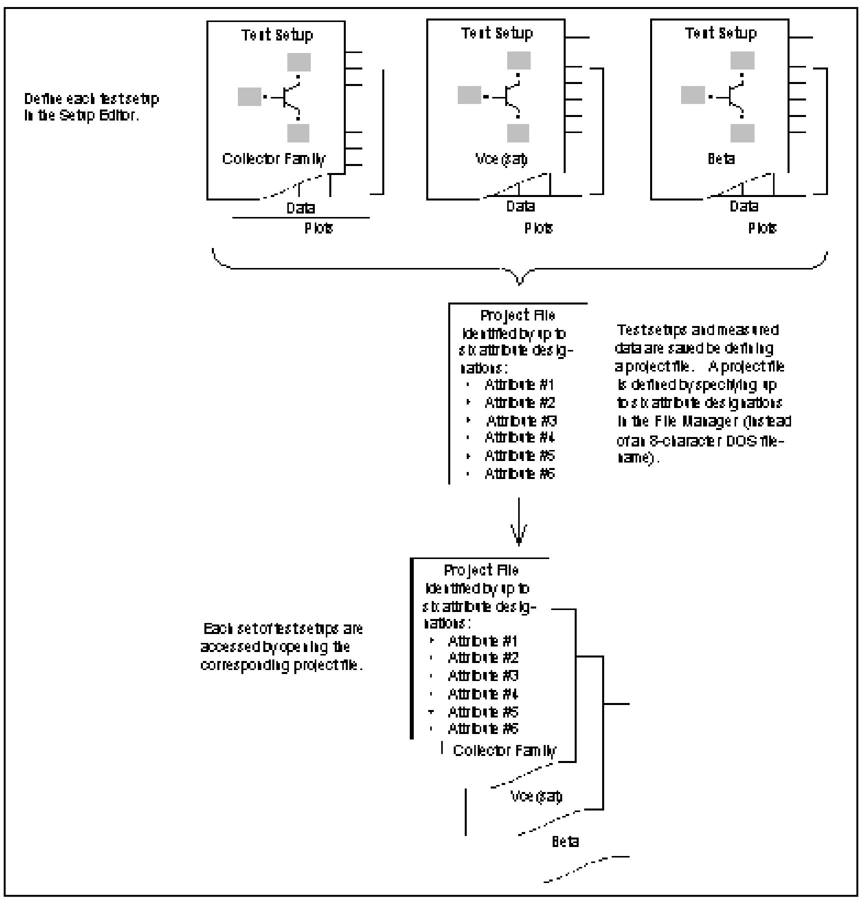


Figure 1-4: One or More Test Setups make up a Single Project File. A Test Setup is only Accessible from the Project File or Temp File in which it was created.

Test Setup Applications

Depending upon the instrumentation under control, 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 across multiple dies on a wafer. ICS may control a switching matrix. By using the scan level as 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 switching topology that identifies a unique wafer location.

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, and
4. If applicable, specify a switching topology.

The graphic environment of the Setup Editor completely eliminates the need for programming of any sort!

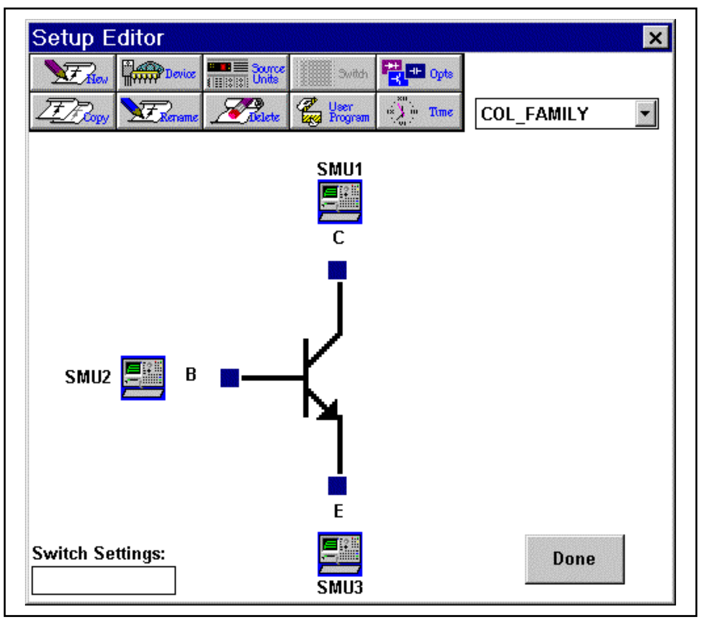


Figure 1-5: The Setup Editor is Used to Create Test Setups

Project Files

A project file is 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 three 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.
3. A project file stores the Instrument Driver selection(s) required to exe-cute the test setup(s).

* When a project file is opened, the necessary instrument drivers are automatically connected to ICS. Any of the defined test setups can be executed immediately. Thus, it is important to make sure when transporting project files between two systems that both systems have the same drivers available.

Creating Project Files

In a conventional DOS environment, files are created by specifying an eight character name followed by a three-character extension. This naming scheme is lim-ited in sophistication and intuition especially when trying to organize a large quantity of files. When trying to organize hundreds of files, you are usually forced to resort to some sort of encrypted file naming system. Such a system is often confusing and hard to manage.
ICS includes a file management system that allows you to create a project file without the limitations of a DOS environment. This method is quite versatile because it allows you to identify project files in a way that is more descriptive of the data than conventional DOS naming schemes.

* In ICS, a project file is not defined by specifying a filename; instead, project files are defined by specifying the contents of up to six project file attribute fields displayed in the File Manager.

After the attribute fields are designated, ICS automatically assigns a DOS file-name and locates the file in a network of directories and sub directories. This function is entirely transparent to the user.

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

Designating Project File Attributes

* The function of the File Manager is similar to the function of a combination lock. A project file is identified by specifying up to six project file attributes. Each unique combination of attribute designa-tions represents a unique project file. The File Manager attribute fields are hierarchical in nature. This means that the first attribute field must be designated before the second, and the second before the third, the third before the fourth, and so on. Each designated attribute field adds another layer of complexity to the database structure.

When saving a project file for the first time, specify a designation in the ATTRIBUTE #1 field. The ATTRIBUTE #1 designation can be any alpha-numeric string up to 23 characters in length. Hit the TAB key and advance to the ATTRIBUTE #2 field. The ATTRIBUTE #2 designation can be any alpha-numeric string up to 12 characters in length. After the ATTRIBUTE #2 designation is specified, TAB to the ATTRIBUTE #3 field and specify an-other alpha-numeric string. Continue specifying the next attribute field until all the fields are designated or until you have designated the number of fields intended.

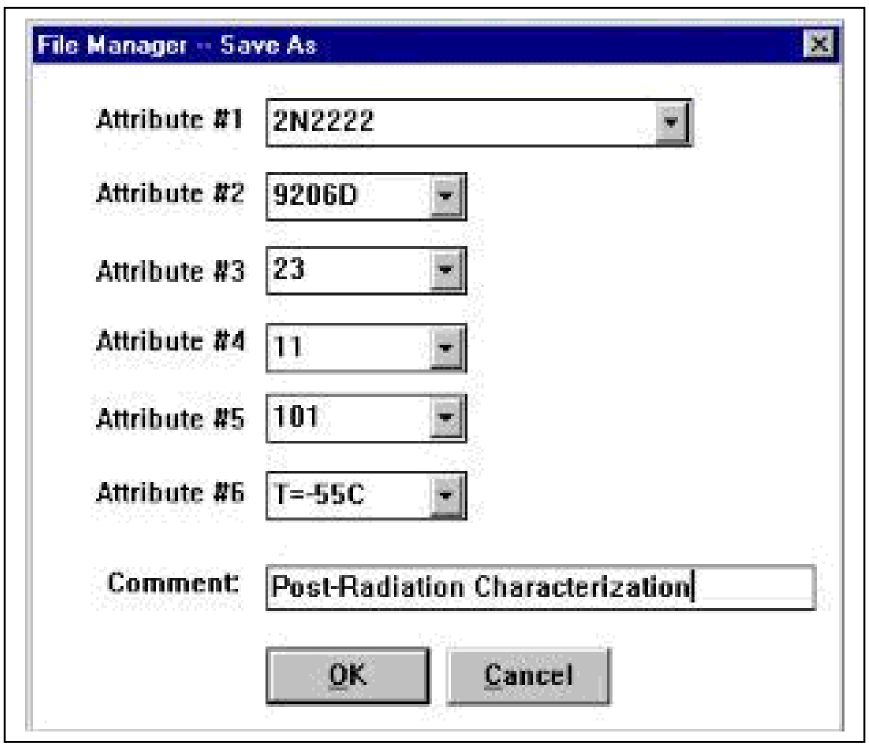


Figure 1-6: A project file is saved by specifying up to six project file attributes in the File Manager. This provides greater versatility than the limitations of specifying a single DOS filename.

Project files are opened by specifying the same combination of file attributes designated when the project file was saved. Selecting an attribute designation will display the corresponding attribute designations in the next attribute field. For example, selecting a single ATTRIBUTE #1 designation may display three designations in the ATTRIBUTE #2 field. Selecting a single ATTRIBUTE #2 designation may display four designations in the ATTRIBUTE #3 field. This process may continue up to the ATTRIBUTE #6 field. Once a designation is selected in the ATTRIBUTE #6 field, the project file is fully identified. Selecting OK in the File Manager will open the project file and display the corresponding test setups at the bottom of the desktop. The diagram shown on the following page outlines a small database of project files. Any possible path through the database attribute designations represents a unique project file. In the diagram shown below, the attribute path is fully displayed for two project files. One of the project files includes the results of a 2N2222 diode (S/N 3) tested at -55C, and the other includes the results of the same device tested at ambient temperature. The File Manager displayed in the diagram shows the attribute designations selected for the -55C project file.

Configuring the Project File Storage Mode

* The project file storage mode designates the maximum number of attribute fields that are accessible in the File Manager.

ICS is shipped with the project file storage mode set to "Wafer". In Wafer Mode, project files are designated according to a six-level hierarchy. This means that all six attribute fields are active. The six fields are labeled ATTRIBUTE #1 through ATTRIBUTE #6. Alternatively, the project file storage mode can be set to "Package". In Package Mode, project files are designated according to a three-level hierarchy. Only the ATTRIBUTE #1, ATTRIBUTE #5, and ATTRIBUTE #6 fields are active.

* When ICS is shipped, the six File Manager attribute fields are labeled ATTRIBUTE #1 through ATTRIBUTE #6 as a default. Alternatively, the File Manager attribute field labels may be customized by the user.

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*.

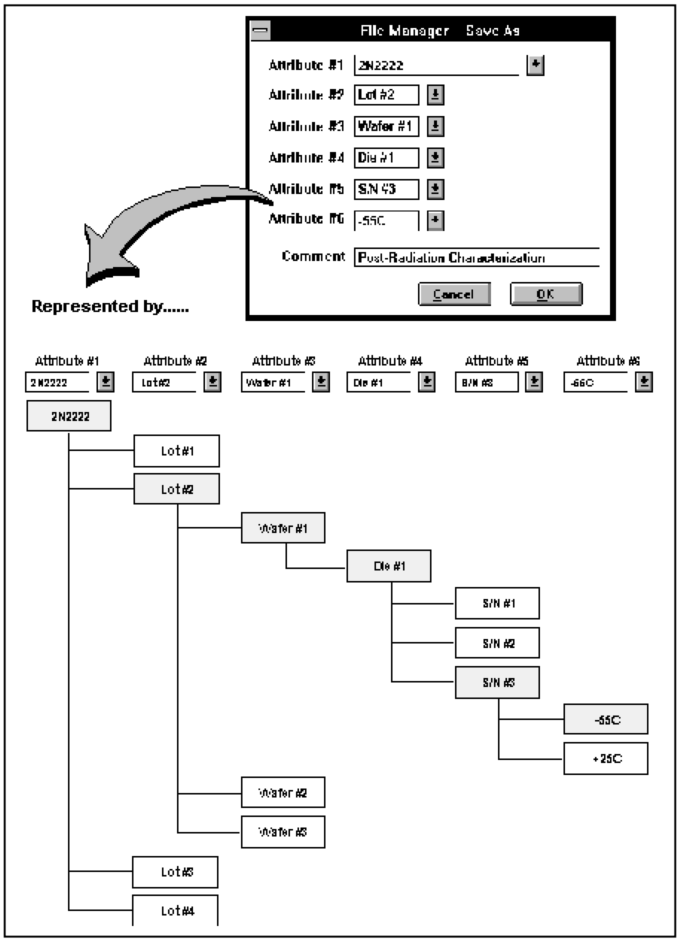


Figure 1-7: An example of a small database of project files. Selecting an attrib-ute designation in the File Manager will display the corresponding attribute designations in the next attribute field. The attribute designations for two project files are fully displayed: S/N3 tested at -55C and +25C. The -55C project file is configured in the File Manager fields shown above.

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 be a database that includes 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. Though this measurement scheme resulted in twelve project files, you don't have to define the three test setups twelve times! In fact, you only have to define the test setups once. Before creating the first project file, make certain that the project file storage mode is set to "Package". If necessary, refer to Chap-ter 3, *Setting the File Manager Options*. Create the test setups and measure the first device at one of the three temperatures. (ICS includes a Sequence Editor that can be used to define a sequence of test setups. A test setup sequence can be executed with a single click of the toolbar MEASURE button. Refer to Chapter 4, *Executing Multiple Test Setups*.) After the data is returned, click the SAVE AS toolbar button and specify a ATTRIBUTE #1 designation, such as "2N2222". Hit the TAB key to advance to the ATTRIBUTE #5 field and specify a designation. Advance to the ATTRIBUTE #6 field and specify the temperature. If desired, comments may be designated in the COMMENTS field. Click the File Manager OK button to accept the project file designations and save your results.

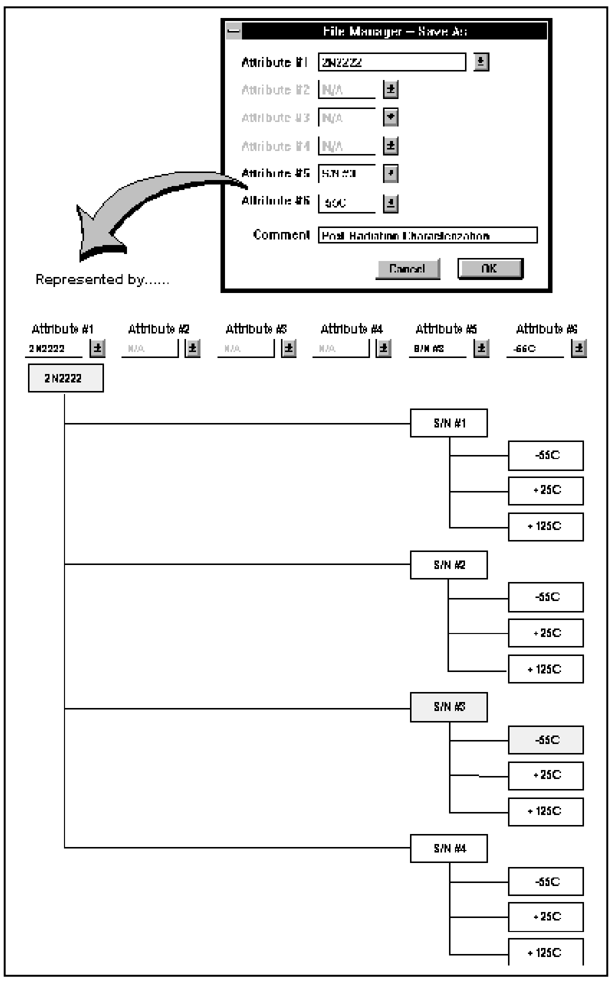


Figure 1-8: An Example of the Project File Designations for 12 Project Files. Each Project File Represents 1 of 4 Devices Tested at 1 of 3 Temperatures.

Expose the device to the second temperature and re-execute the three test set-ups 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 autostore function has not been activated in the Output Data control). Click the SAVE AS toolbar button and update the tem-perature specification in the File Manager ATTRIBUTE #6 field. 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 se-lect 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.

Single Measurement/Multiple Devices

Consider an application in which you are required to measure VCE(sat) curves for a sampling of die locations on a given wafer. We'll assume that you will use a switching matrix in combination with a manual wafer prober. One possible organization of your data is to locate all of the measurements for a given wafer in a single project file. The final result will be a project file for every tested wafer. In this case each test setup within a project file will configure the same device measurement, VCE(sat), but each test setup will specify a unique switching topology representing a different wafer location, or die. In this example the project file storage mode will be set to "Wafer". A schematic of the project file designations is shown below. Notice that only three of the six available fields are used.

Before creating the first project file, make certain that the project file storage mode is set to "Wafer". Refer to Chapter 3, *Setting the File Manager Options*, if necessary. Create the test setups and measure the first wafer. (ICS includes a Sequence Editor that can be used to define a sequence of test setups. A test setup sequence can be executed with a single click of the toolbar MEASURE button. Refer to Chapter 4, *Executing Multiple Test Setups*.) After the data is returned, click the SAVE AS toolbar button and specify an ATTRIBUTE #1 designation in the respective field, such as "Vce(sat) Wafer Tests". Hit the TAB key to advance to the ATTRIBUTE #2 field and specify a ATTRIBUTE #2 designation. Advance to the ATTRIBUTE #3 field and specify a designation. If desired, comments may be designated in the COMMENTS field. Click the File Manager OK button to accept the project file designations and save your results.

Change wafers and re-execute the test set-ups 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. Click the SAVE AS toolbar button and update the File Manager ATTRIBUTE #3 designation. Click the File Manager OK button to accept the project file designations and save your results. You have just created a second project file.

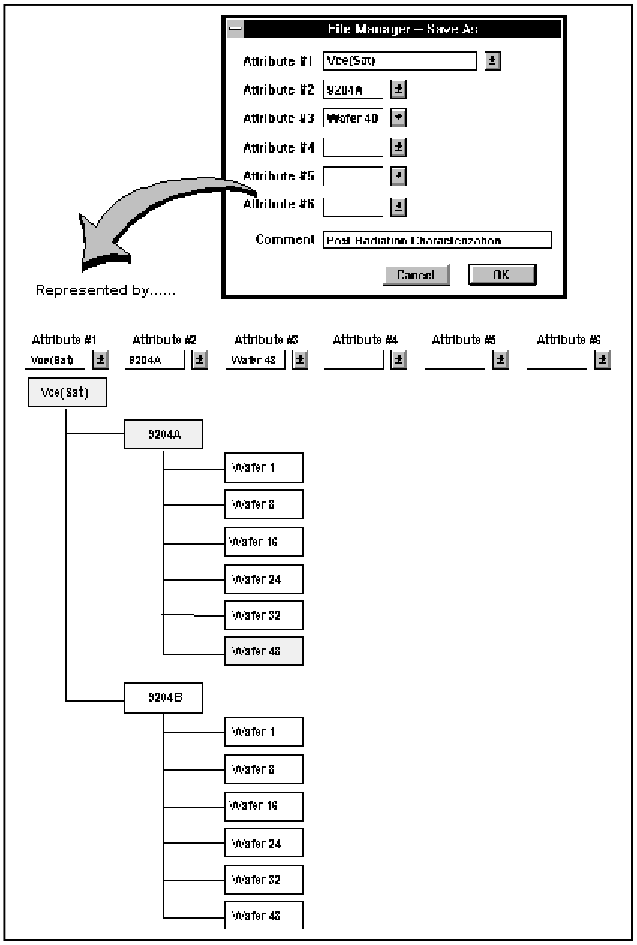


Figure 1-9: An Example of the Project File Designations for 12 Project Files. Each Project File Represents 1 of 48 Possible Wafers From 1 of 2 Tested Lots.

The ICS Initialization File (ICS35.INI)

The ICS.INI file is stored in the Windows root directory. The ICS35.INI file is created during the installation of the ICS Program Disk. The file is updated automatically during the installation of any Driver Library Disk or as a result of certain procedures executed within ICS. The ICS35.INI file is read by ICS each time ICS is started or a new driver is connected. The specifications defined in the ICS35.INI file DO NOT have to be specified each time ICS is started.

The ICS.INI file will be updated when:

1. The communications hardware installed in your computer changes.
2. The communications address of any instrument changes.
3. The available source units of a particular instrument change.
4. Any plug-in source unit modules are installed into different mainframe slots.
5. ICS is customized by changing the workspace controls.

The ICS.INI file is divided into four sections:

1. Environment Specifications

The specifications included in this section specify the locations of the ICS directories and sub-directories. These specifications are defined by the user during the installation of the ICS 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 ICS Program, but they can be reconfigured while running ICS. If you replace the communications card installed in your computer, you can update the ICS35.INI file without the need to re-install ICS. 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 an instrument driver is connected, any corresponding GPIB addresses are set automatically.

Some measurement systems consist of stand-alone SMUs or plug-in SMU mod-ules. The first time that an instrument driver of this type is connected to ICS, the hardware identity of each available SMU must be mapped to a ICS SMU designation. Once defined, the SMU map is written to the ICS35.INI file and never has to be designated again until the available SMUs change, or the SMU modules are installed into different mainframe slots (if applicable).

For detailed instructions explaining how to designate the GPIB address(es), or how to map SMU instrument identities to ICS SMU designations, refer to *Specify the GPIB Address(es)* in the Driver Library chapter of this manual corresponding to your instrument.

4. Workspace Controls

The workspace controls are those specifications that define the customized fea-tures of ICS, 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 ICS 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 ICS desktop. ICS 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 en-titled "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 spread-sheet 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.

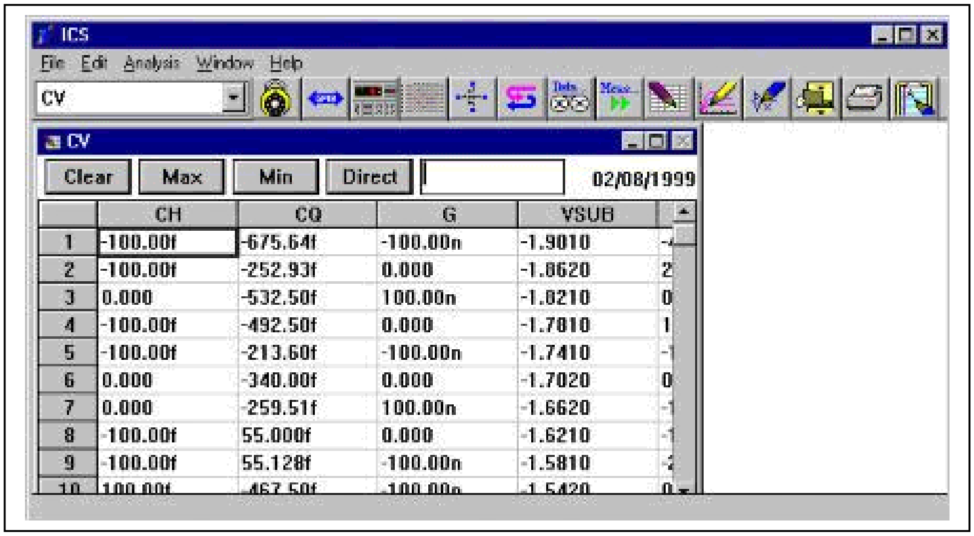


Figure 1-10: An Example of an ICS Data Window Spreadsheet.

* 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.

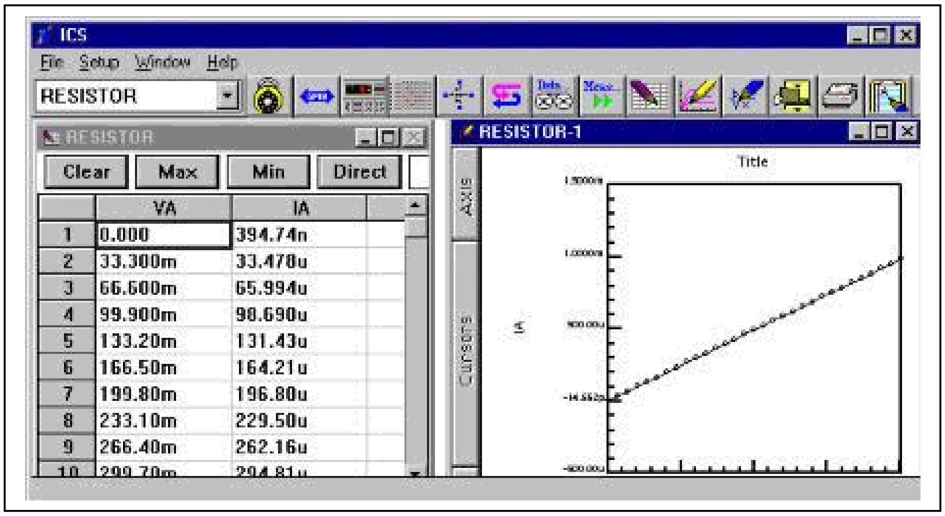


Figure 1-11: A Plot Window is Dynamically Linked to the Corresponding Data Window Spreadsheet.

The Menu Bars

ICS 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

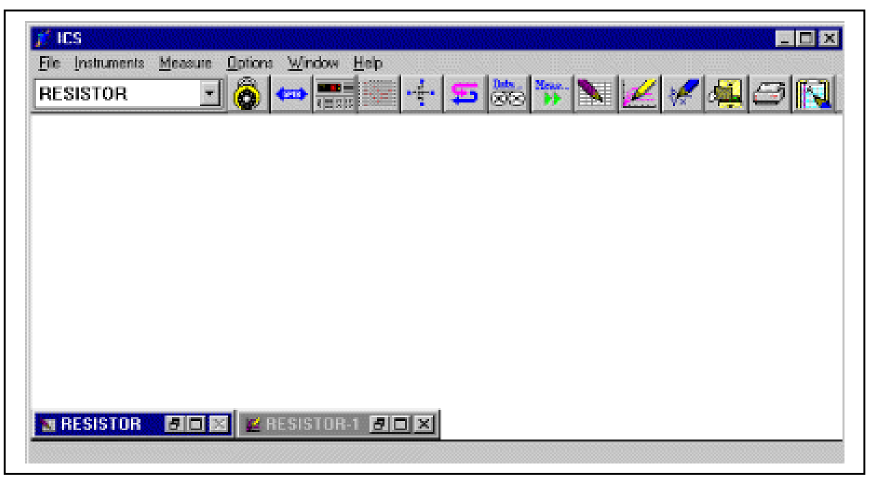


Figure 1-12: The Measurement Mode Menu Bar.

When ICS is started, ICS 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 com-pleted 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.

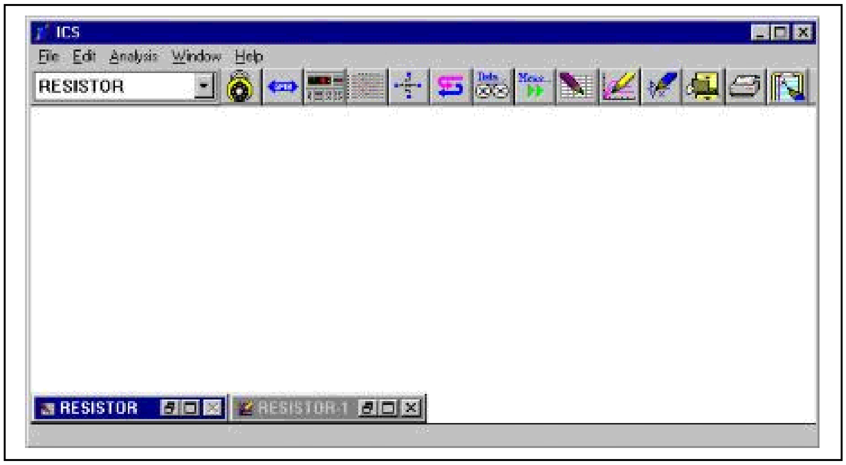


Figure 1-13: The Data Mode Menu Bar.

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, Op-tions, 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.

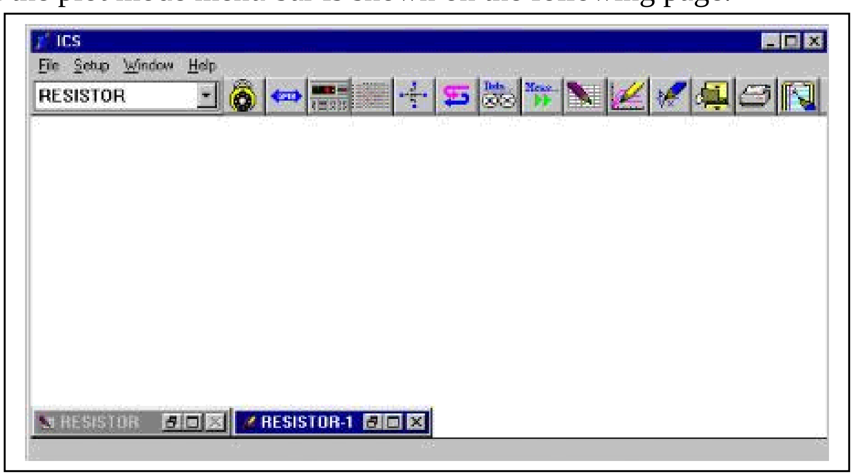


Figure 1-14: The Plot Mode Menu Bar.

The Toolbar and Status Bar

ICS 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 ICS. The *default* display configurations can also be changed in the Work-space Controls dialogue box.

How to Toggle the Toolbar and Status Bar Displays During the Current Execution of ICS:

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 until ICS is restarted. When ICS is restarted, the state of the toolbar and status bar displays will be configured according to the selections designated in the Workspace dialogue box.

How to Toggle the Default Toolbar and Status Bar Displays:

1. From the measurement mode menu bar, selection OPTIONS/WORKSPACE. This will open the Workspace dia-logue box.
2. In the Tools section of the dialogue box, configure the toolbar and status bar switches. If a switch is selected, the corresponding tool will default to ON.
3. This will close the Workspace dialogue box. The selections just configured will be take affect the next time ICS is started.

The Toolbar

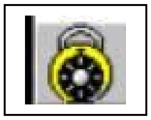
The toolbar is designed to provide easy access to the features most likely needed while using ICS. Each of the toolbar features are accessible through the appropriate drop-down list located in one of the menu bars, but the toolbar allows you to access the corresponding feature with a click of a single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

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

Setup Window

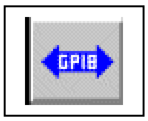
The setup window displays the active test setup. Click the corresponding scroll arrow to display a list of available test setups. The active test setup can be changed by selecting one of the available options.

Codeword Setup



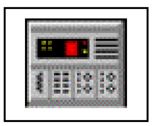
This button will open the Password Setup dialogue box. The Password Setup dialogue box is used to activate the ICS software. The codeword is required to activate the ICS software. Please refer to the ICS Installation Guide for instructions to set and reset your password.

Communications Setup



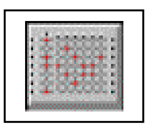
This button will open the Communications Setup dialogue box. The Communications Setup dialogue box is used to specify the communications type and associated parameters to ICS.

Connect Instruments



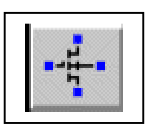
This button will open the Connect Instruments dialogue box. The Connect Instruments dialogue box is used to connect an instrument driver to ICS.

Connect Switching Matrix



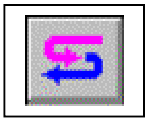
This button will open the Switch Application's main window. The Switch Application is used to setup switch settings and link the settings to ICS.

Setup Editor



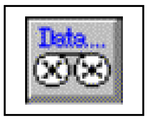
This button activates the Setup Editor. The Setup Editor will initially display the active test setup.

Sequence Editor



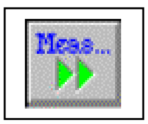
This button will open the Sequence Editor dialogue box. The Sequence Editor dialogue box is used to switch ICS between Single Test or Multiple Test mode as well as define a test setup sequence.

Global Controls



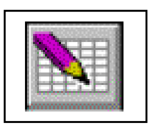
This button will open the Global Controls dialogue box. The Global Controls dialogue box is used to control the Auto-Store and Auto-Export functions as well as the mode of the temporary file.

Measure



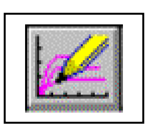
The measure button actuates the Measurement Remote Control dialogue box which is used to initiate all type of measurement.

Data View



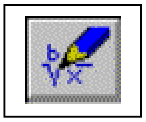
This button opens the Data View Setup dialogue box. The Data View Setup dialogue box is used to specify what data vectors returned from the instrument driver are to be displayed in the data spread sheet and save to the project file.

New Plot



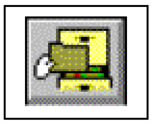
This button opens the Setup Data dialogue box. The Setup Data dialogue box is used to create a new plot.

Transform Editor



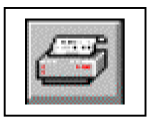
This button activates the Transform Editor. The Transform Editor is used to add new data vectors to a data window spreadsheet. These calculated data vectors are analytical expressions of measured data vectors.

Save As



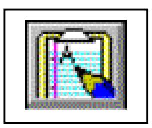
This button opens the File Manager dialogue box allowing the user to define a new project file. A project file is defined by specifying up to six project file attributes.

Active Window to Printer



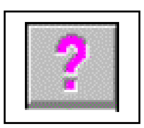
This button is used to send the contents of the active window to the printer.

Active Window 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 ICS desktop. The status bar displays messages throughout the operation of ICS. These messages either indicate what feature or function of ICS is currently active, or the messages prompt the user for an action. If in doubt about what to donext, glance at the status bar.

Engineering Units

ICS 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, ICS 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", ICS multiplies 250 by 10^{-6} to arrive at the numerical value of the field entry. A list of the engineering units recognized by ICS is summarized in the table below.

Notation	**Multiplier**
f	$x10^{-15}$
p	$x10^{-12}$
n	$x10^{-9}$
u	$x10^{-6}$
m	$x10^{-3}$
K	$x10^{3}$
M	$x10^{6}$
G	$x10^{9}$

Table 1-1: Engineering Notation Recognized by ICS

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.

Information Stored in a Test Setup

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

1. A Test Setup Defines the Instrument Setup Configuration of Each Source Unit:

ICS 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. ICS 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, ICS 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 Ids1, Ids2, Ids3, etc. Each data vector corresponds to a single curve. ICS 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 ICS).

4. A Test Setup Defines the Switching Topology:

When a switching matrix is integrated with the measurement system, the switching topology name is part of the test setup definition. This allows you to create a group of test setups that are distinguished by varying switch configurations.

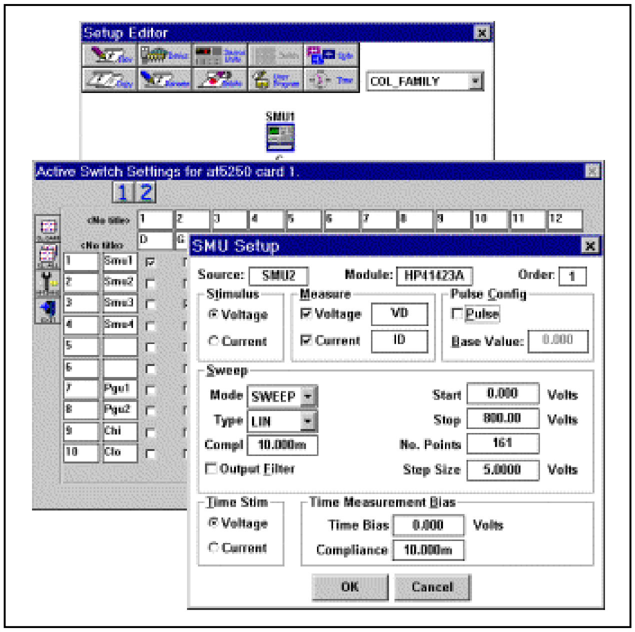


Figure 2-1: The test setup stores instrument setup conditions, documents the instrument/DUT connections, stores what data is returned and how it is labeled, and stores the switching topology name if a switching matrix is present.

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 ten test setups in a single project file. If this limit is exceeded the stability of the operating system may deteriorate.

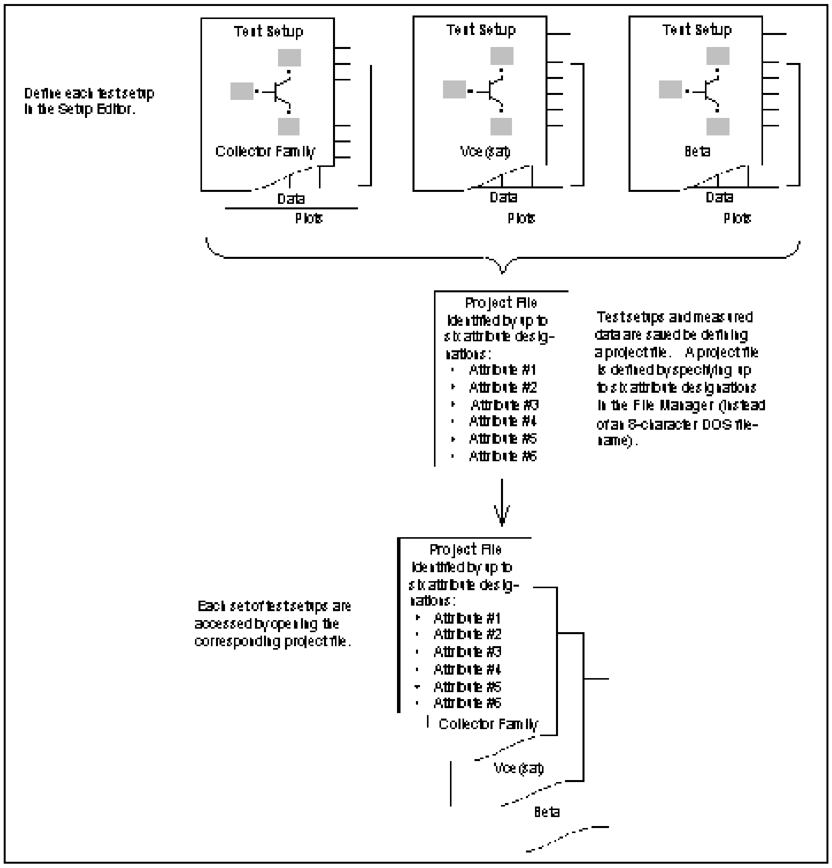


Figure 2-2: One or More Test Setups make up a Single Project File. A Test Setup is only Accessible from the Project File or Temp File in which it was Created.

* 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 occurs when you use the Autostore feature, but this is a higher function of ICS covered elsewhere in this manual. In general, the latter point is true.

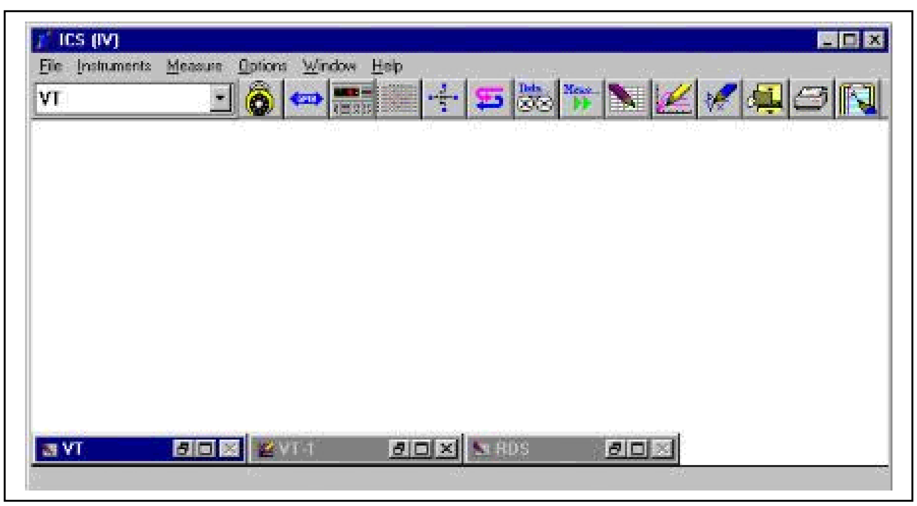


Figure 2-3: Test Setups Corresponding to a Project File are Displayed Along the Bottom of the Desktop.

Test Setup Applications

Depending upon the instrumentation under control, 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. ICS may control a switching matrix to allow the use of a probe card. By using the scan level as attributes 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 switching topology that identifies a unique device. Consult your ICS automation guide for more information.

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:

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, and
4. If applicable, specify a switching topology.

The graphic environment of the Setup Editor completely eliminates the need for programming of any sort!

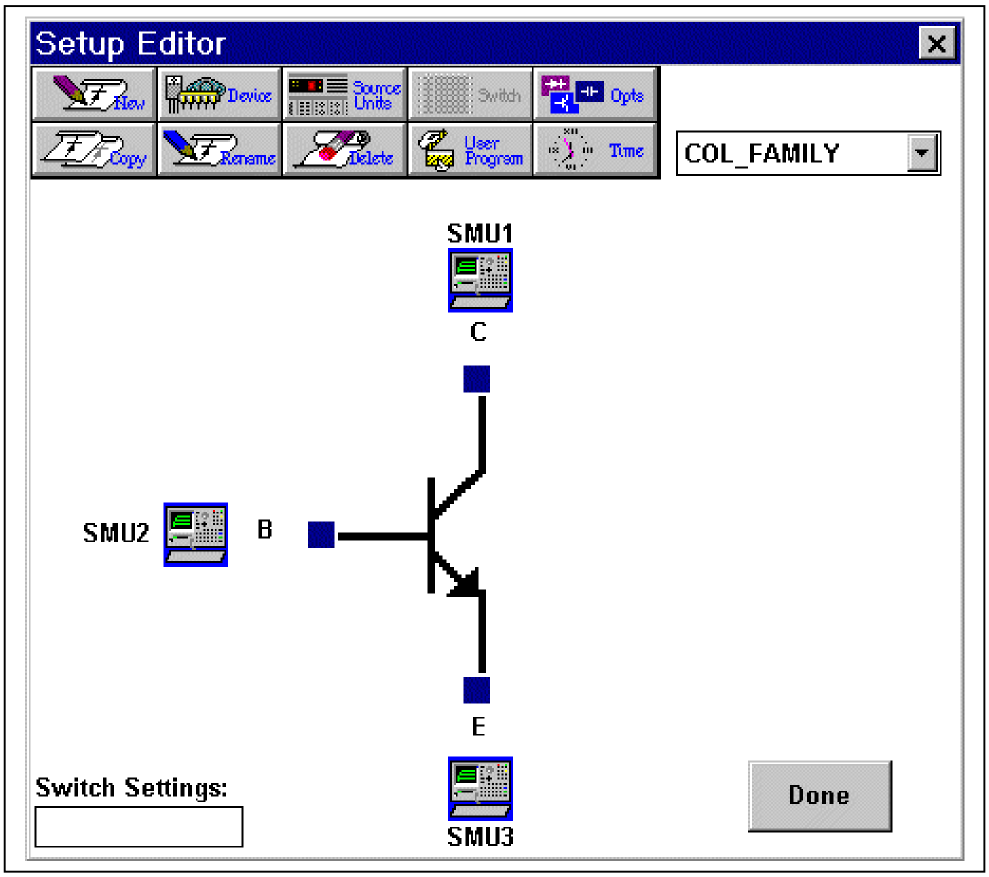


Figure 2-4: The Setup Editor is Used to Document Instrument/DUT Connections and Configure Instrument Setup Conditions.

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 in-structions specific to a single device measurement; therefore, a project file may contain numerous test setups.

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:

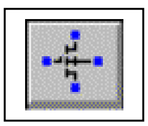
1. Open the Setup Editor. Click the NEW button and specify a test setup name at the prompt.
2. Click the DEVICE button and choose a device schematic that corresponds to the Device Under Test (DUT).
3. From the Setup Editor, open the Source Units dialogue box by selecting the SOURCES button. Designate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

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.

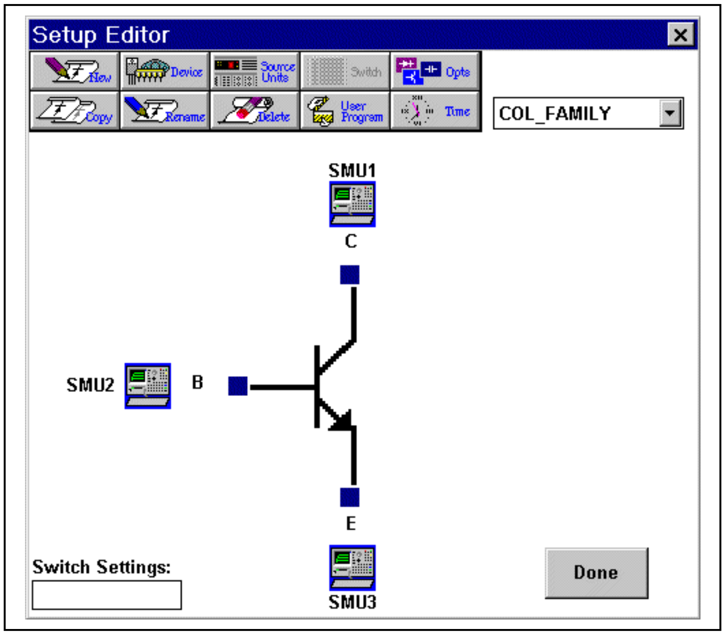


The Setup Editor can be opened by clicking the Setup Editor tool bar button.

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 dialogue 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.

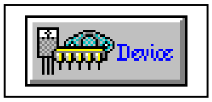
* By selecting the appropriate device schematic, the user can designate the hardware connections between the instrument and the device. This allows the user to create a visual record of the hardware configuration required for the corresponding test setup.

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 presen-tation 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. ICS 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 con-figured 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.

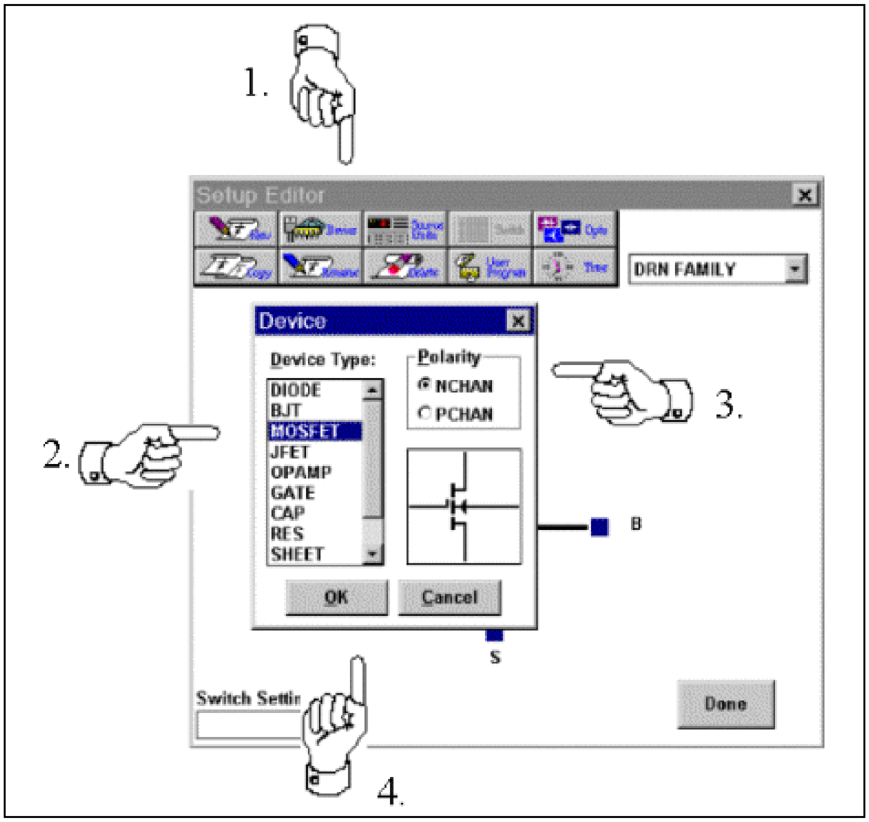


Figure 2-5: How to Select the Device Schematic.

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 twelve 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.

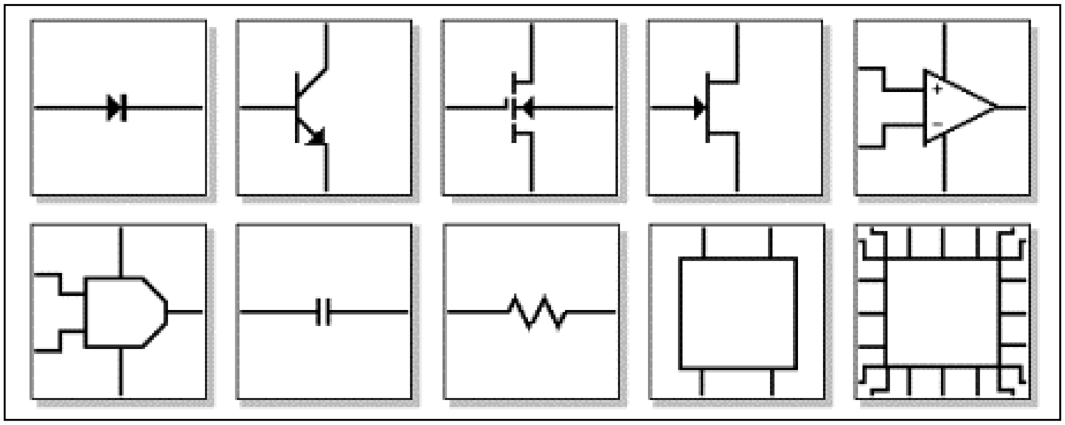


Figure 2-6: There Are a Total of Twelve Device Schematics That Can Be Selected From the Device Dialogue Box.

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.

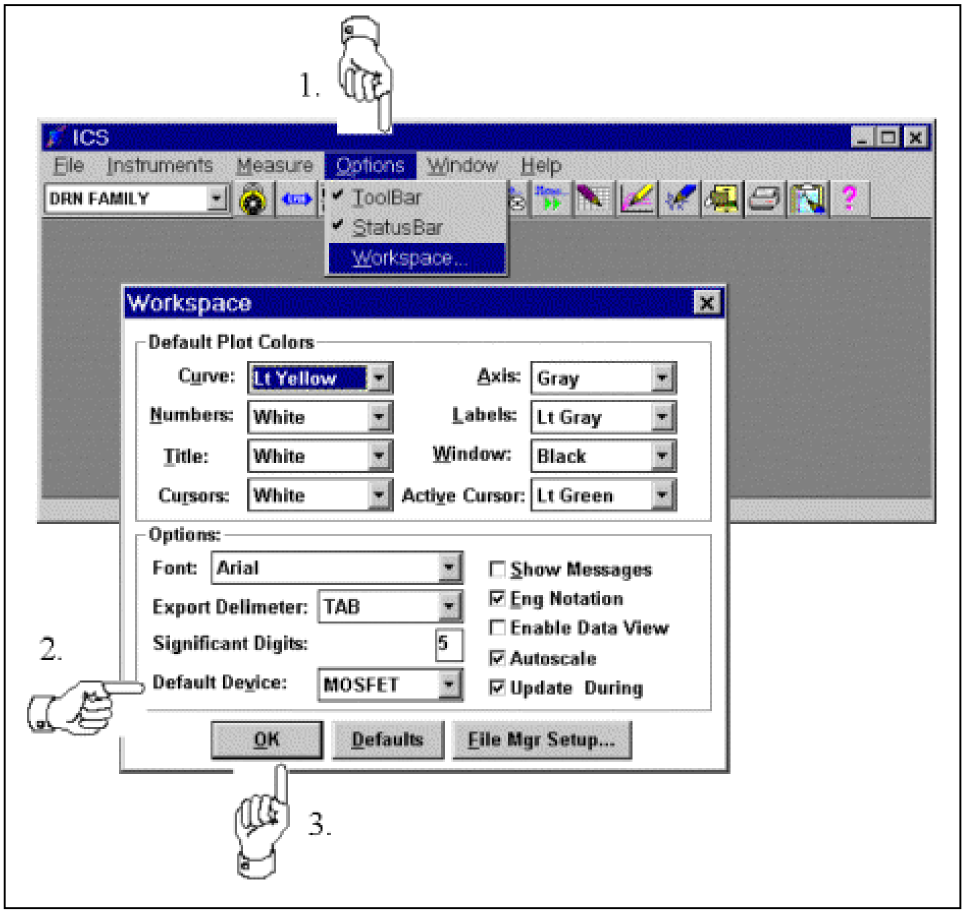


Figure 2-7: How to Specify the Default Device Schematic

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 des-ignation.
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. ICS 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.

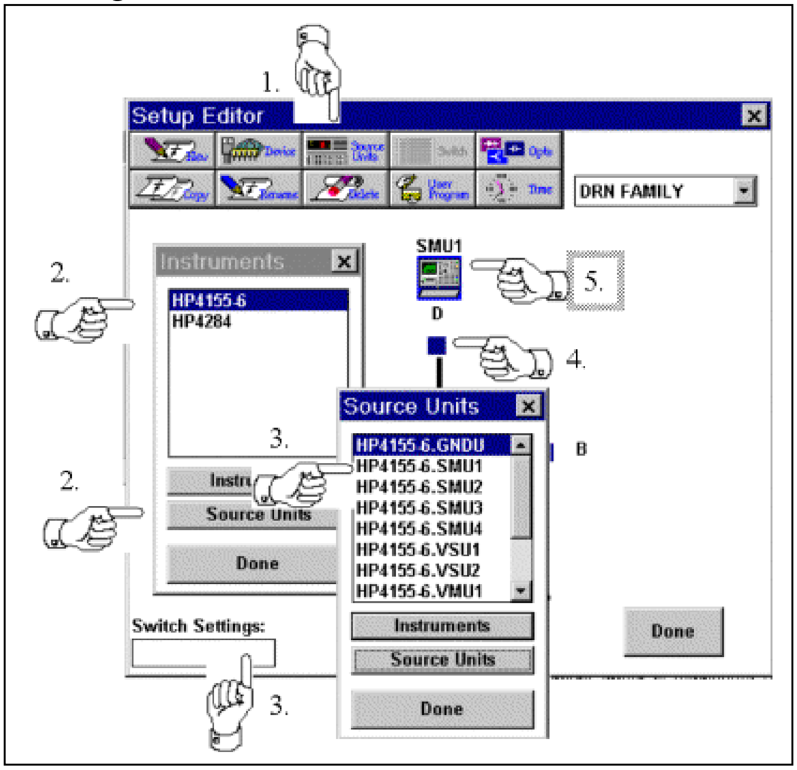
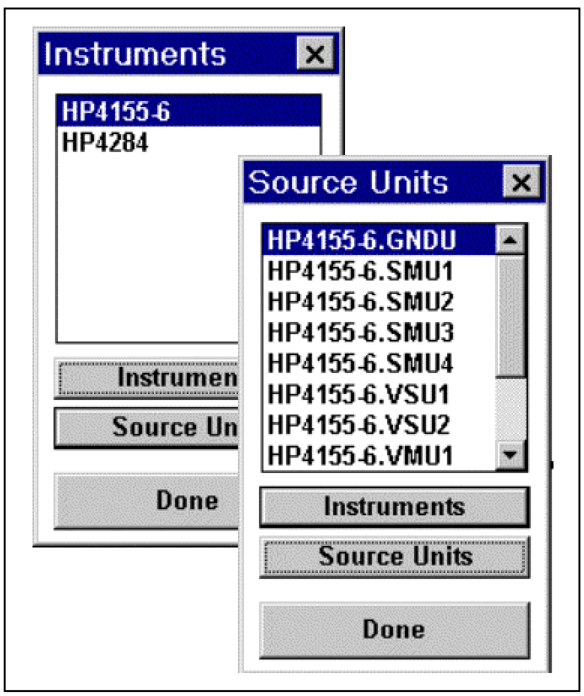


Figure 2-8: How to Designate a Connection Between the Instrument and the DUT.

How to Designate the Instrument/DUT Connections:

1. Open the Source Units dialogue box from the Setup Editor.
a. If more than one instrument driver is connected to the project file, the Source Units dialogue box will display a list of corresponding instrument designations. Continue with Step #2.
b. If only one instrument driver is connected to the project file, the Source Units dialogue box will display a list of available instrument supplies. Continue with Step #3.

2. Open the menu of available source units by highlighting the corresponding instrument designation and clicking the SOURCE UNITS button at the bottom of the dialogue box. The source units list can also be displayed by double clicking on the instrument designation. Source units are unique to each design of test instrumentation. For this reason, the available source units depend upon the connected instru-ment driver(s).
3. Highlight the desired source unit.
4. Designate a connection by clicking on the blue pad next to the desired schematic pin. An instrument icon will appear above the connection.
5. 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.



When more than one instrument driver is connected to the active project file, the Source Units dialogue box will display the corresponding instrument designations. Highlight the desired instrument designation and click the SOURCE UNITS button at the bottom of the dialogue box to display a list of the corresponding source units. When only one instrument driver is connected to the active project file, the Source Units dialogue box will automatically display a list of the available source units.

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. ICS will display a message box asking you to confirm your desire to undesignate the connection. Click YES to complete the procedure.

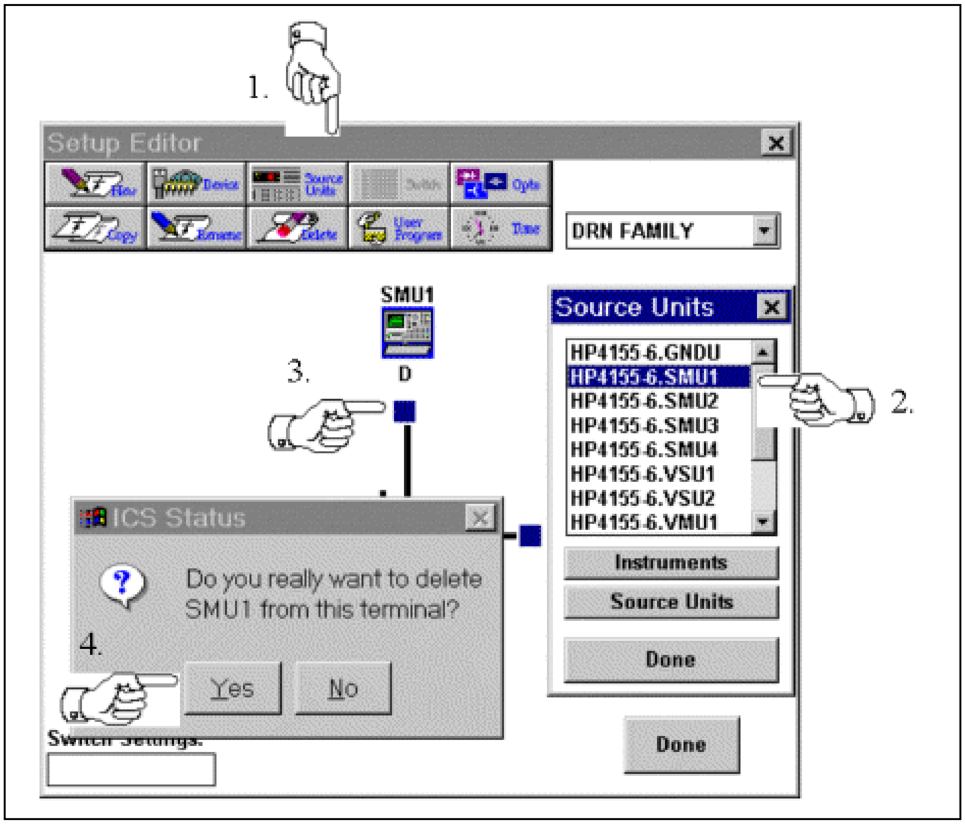


Figure 2-9: How to Undesignate an Instrument/DUT Connection

How to Remove an Instrument/DUT Connection

1. Click the Setup Editor SOURCES button. This will open the Source Units dialogue box. If more than one instrument driver is connected, select the appropriate instrument and click the SOURCE UNITS button at the bottom of the Source Units dialogue box. If only one instrument driver is connected, the available source units and instrument supplies will be displayed immediately.
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. ICS will display a message box asking you to confirm your desire to undesignate a connection. Click YES to complete the procedure and restore control to the Setup Editor.

Removing Unlisted Source Units

* It is possible to open a project file that was created with a hardware configuration that does not match the configuration of the hardware presently available. This is called a hardware mismatch.

As an example, consider an instrument that consists of plug-in SMU modules. A hardware mismatch occurs when a slot position in the available instrument does not correspond to the same SMU module that was installed in the same slot of the original instrument. When a hardware mis-match occurs, the hardware identities designated in the Instrument Configuration dialogue box will no longer correspond to the designations that were applicable when the test setup was created. In other words, the mismatched SMU labels attached to the device schematic in the Setup Editor no longer correspond to the instrument modules that were used to create the test setup. In some cases, an SMU label may no longer correspond to any source unit.

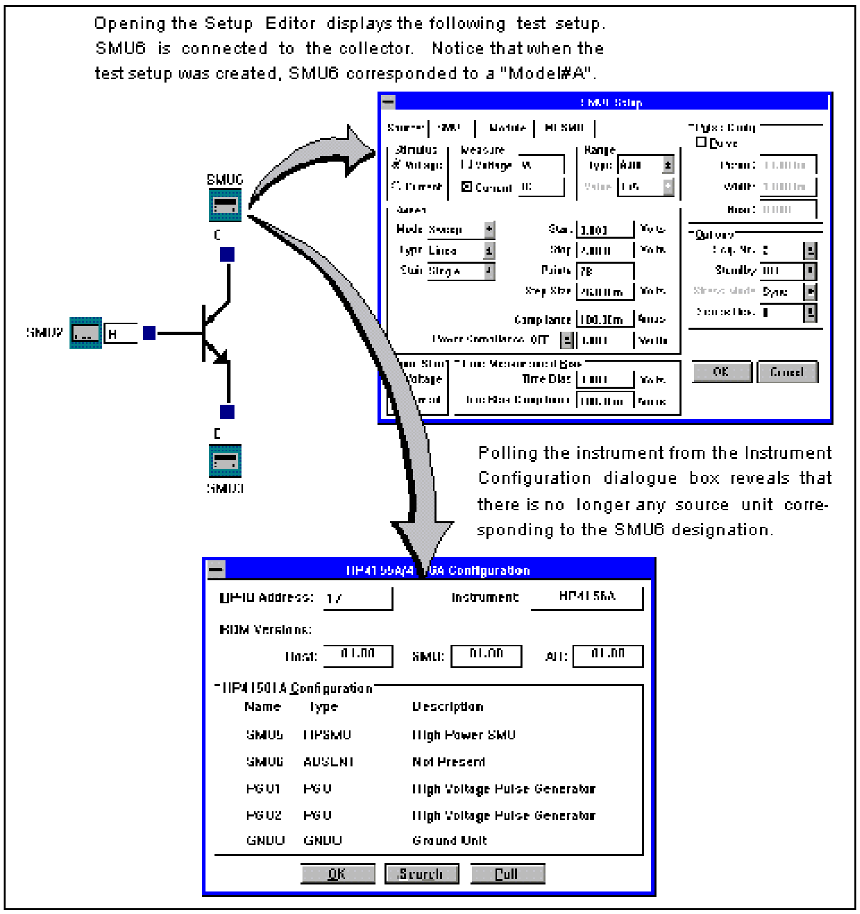


Figure 2-10: The Source Unit Designated in the Test Setup is no Longer Available with the Present Hardware Configuration.

* The Setup Editor display is independent of any hardware designations updated in the Instrument Configuration dialogue box. If the Setup Editor documents a connection between an obsolete source unit and the device, the obsolete source unit must be removed from the Setup Editor device schematic and replaced with an available source unit.

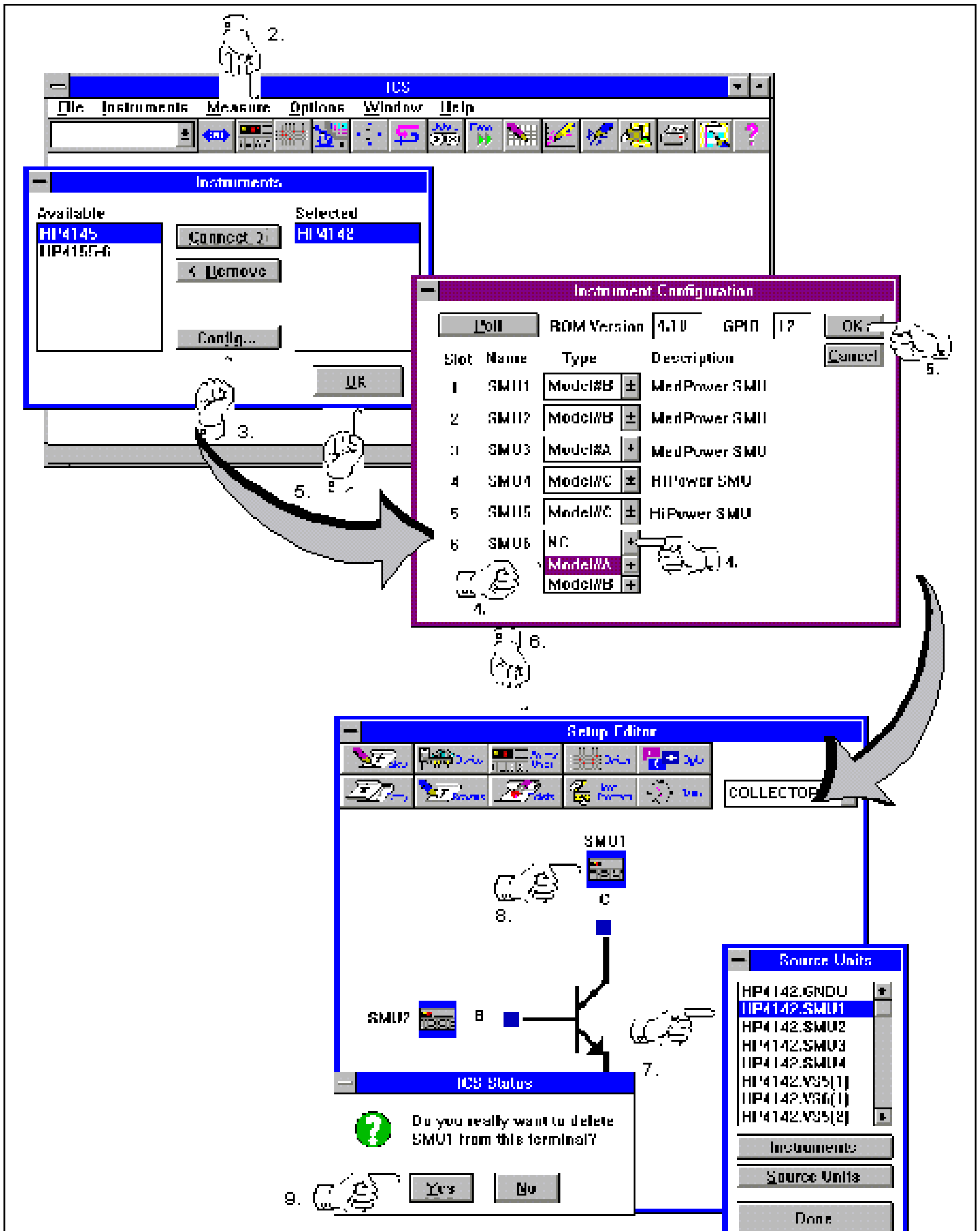
In order to undesignate a source unit connection in the Setup Editor, the identity of the source unit must be selected in the Source Units dialogue box. The Source Units dialogue box is displayed by clicking the Setup Editor SOURCES button. The list of source units displayed in the Source Units dialogue box is based upon the source unit identities designated in the Instrument Configuration dialogue box. If the original source unit is no longer incorporated in the hardware arrangement, the obsolete source unit will not be identified in the Instrument Configuration dialogue box after the instrument is polled. Consequently, the obsolete source unit will not be included in the Source Units list either. The unavailability of the obsolete source unit in the Source Units dialogue box presents a problem when trying to undesignate the original connection. This problem is eliminated by manually editing the Instrument Configuration dialogue box so that the obsolete source unit is designated in one of the SMU fields. After the source unit connection is undesignated in the Setup Editor, restore the edited designation in the Instrument Configuration dialogue box so that it corresponds to the hardware arrangement. Use the following procedure to undesignate a Source Unit Connection when the Source Unit is unavailable.

How to Remove an unavailable Source Unit Connection:

1. Click the SOURCES button in the Setup Editor and confirm that the source unit you wish to remove is not listed in the Source Units dialogue box.
2. Close the Source Units dialogue box and close the Setup Editor. Open the Connect Instruments dialogue box using the corresponding toolbar button. Make sure the correct driver is connected.
3. Open the Instrument Configuration dialogue box by clicking the CONFIG button at the bottom of the Connect Instrument dialogue box.
4. Click the scroll arrow at the right of any SMU field. This will display a list of possible source unit designations. Select the identity of the source unit you wish to remove from the Setup Editor display.
5. Click the OK button to close the Instrument Configuration dialogue box. Click the OK button in the Connect Instrument dialogue box to restore control to the desktop.
6. Open the Setup Editor. Click the SOURCES button to display the Source Units dialogue box. If more than one driver is connected, designate the instrument and click the SOURCE UNITS button at the bottom of the dialogue box. The identity of the source unit you wish to remove will now appear in this list.
7. Designate the obsolete source unit in the Source Units list.
8. Click the blue pad next to the connected source unit you wish to remove.
9. ICS will display a message box asking you to confirm your intention to remove the source unit. Answer YES to remove the source unit.
10. Close the Source Units dialogue box. Close the Setup Editor. Open the Connect Instruments dialogue box a second time.
11. Open the Instrument Configuration dialogue box a second time by clicking the CONFIG button at the bottom of the Connect Instruments dialogue box.
12. Click the POLL button in the Instrument Configuration dialogue box. This will interrogate the instrument and automatically designate the source unit identity of each SMU field in order to match the hardware arrangement. Alternatively, you can manually restore the proper designation in the SMU field edited in Step #4. To manually restore the SMU field, click the scroll arrow to display a list of possible

source unit designations. Select the designation that corresponds to the hardware configuration.

13. Click the OK button to close the Instrument Configuration dialogue box. Click the OK button in the Connect Instruments dialogue box to restore control to the desktop.
14. Return to the Setup Editor and edit the test setup as necessary.



Continued on next page

Figure 2-11A: How to Undesignate a Source Unit Connection When the Source Unit is Unavailable

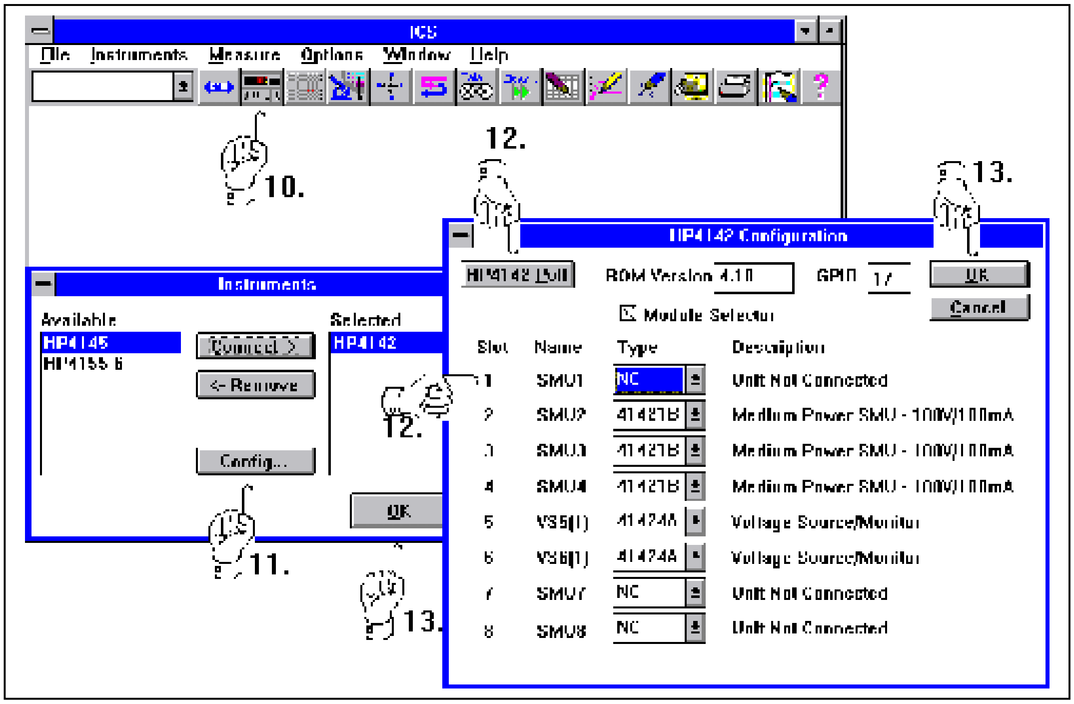


Figure 2-11B: How to Undesignate a Source Unit Connection When the Source Unit is Unavailable

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.

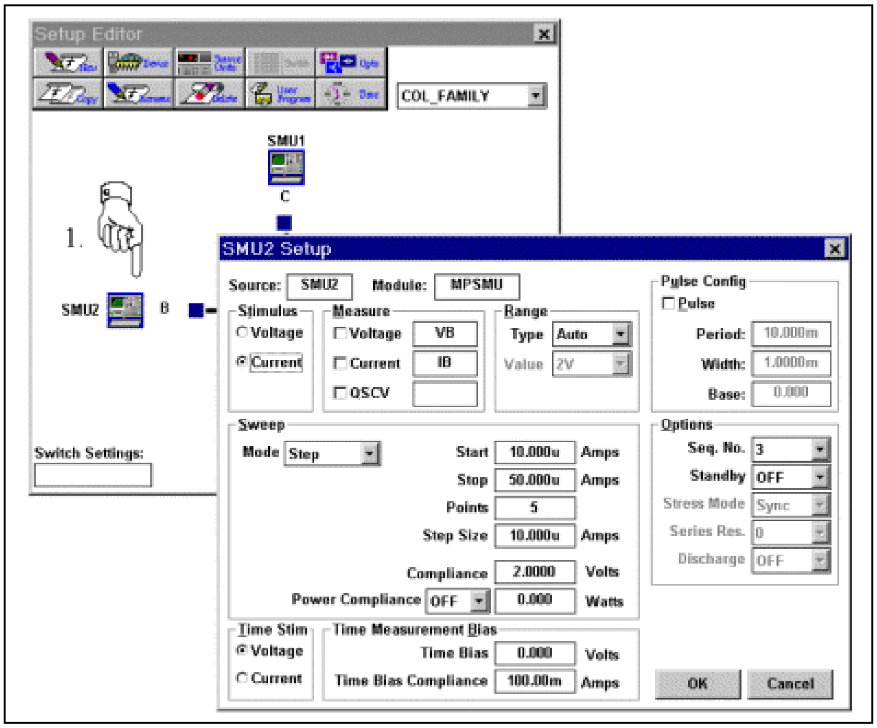


Figure 2-12: How to Display a Setup Dialogue Box When a Single Source Unit is Connected to the Device Schematic Pin.

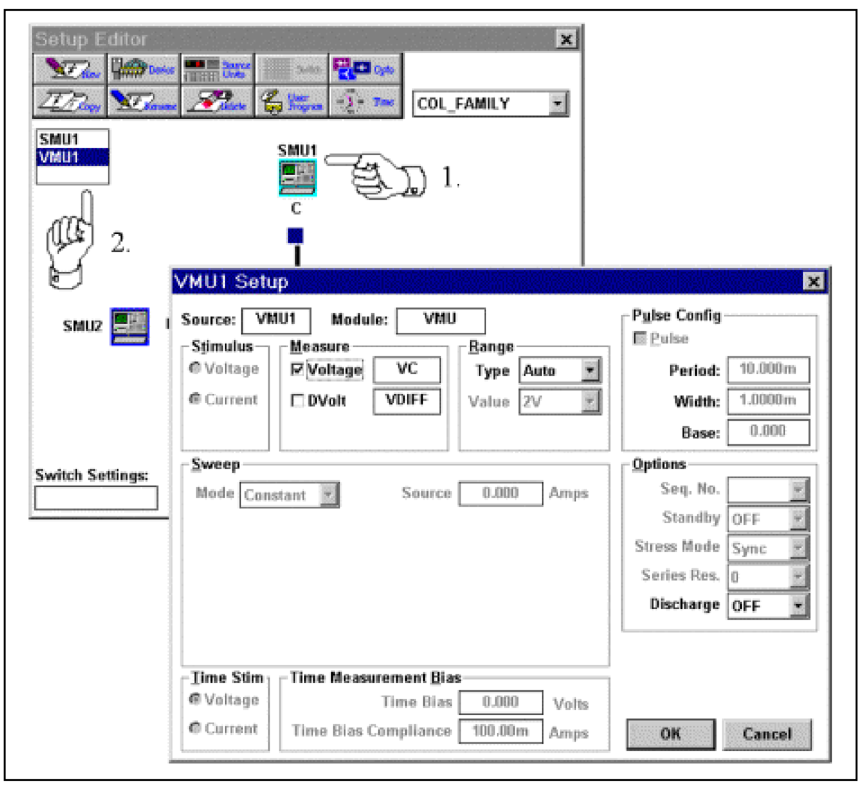
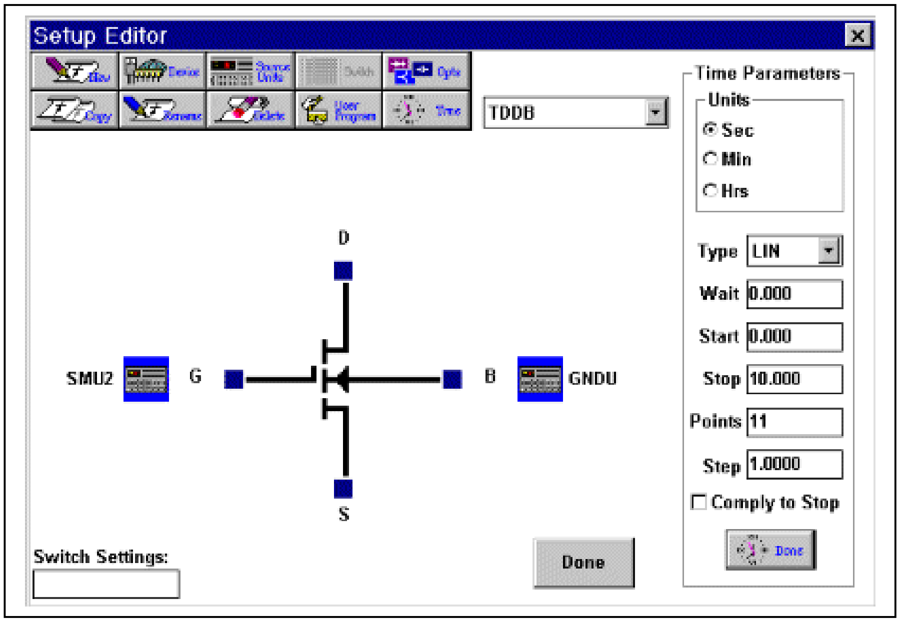


Figure 2-13: How to Display a Setup Dialogue Box When Multiple Source Units are Connected to the Device Schematic Pin.

Creating a Time Domain Test Setup



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 ICS 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.

Sequence Stress mode allows the user to stress the device with a constant bias signal before performing multiple device measurements.

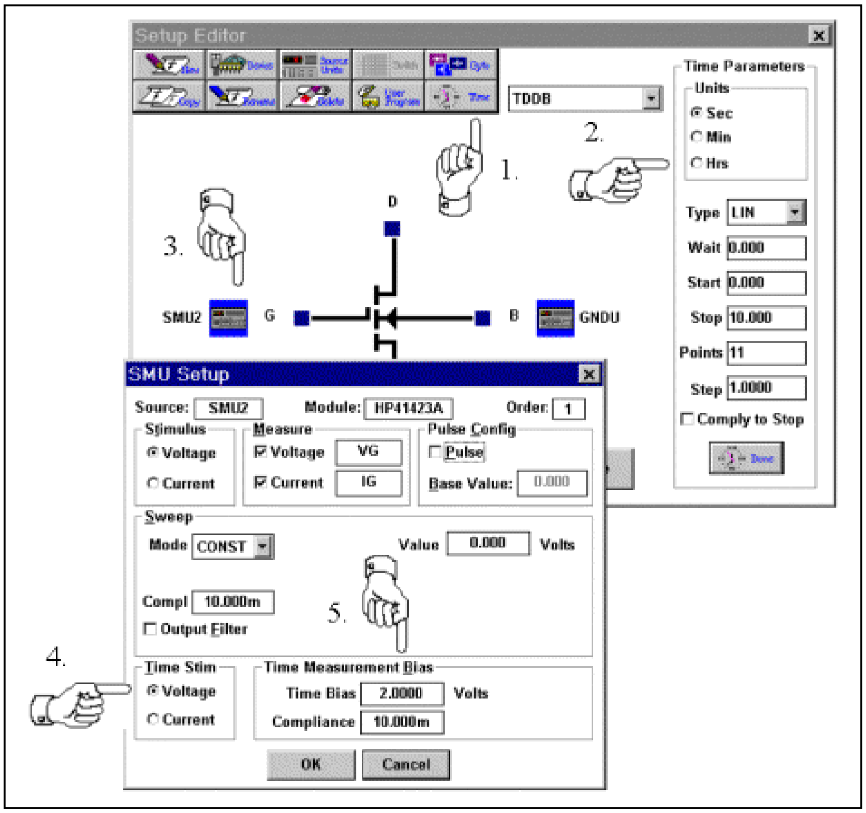


Figure 2-14: Configuring Source Units for ICS Time Measurements.

How to Setup ICS Time Measurement Parameters:

1. Display the ICS Time Measurement controls by clicking on the TIME button.
2. Specify the Bias Delay, Time Measurement, or Sequence Stress 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 or specify the constant stress source values for Sequence Stress 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 configura-tions 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, ICS 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 debiased 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.

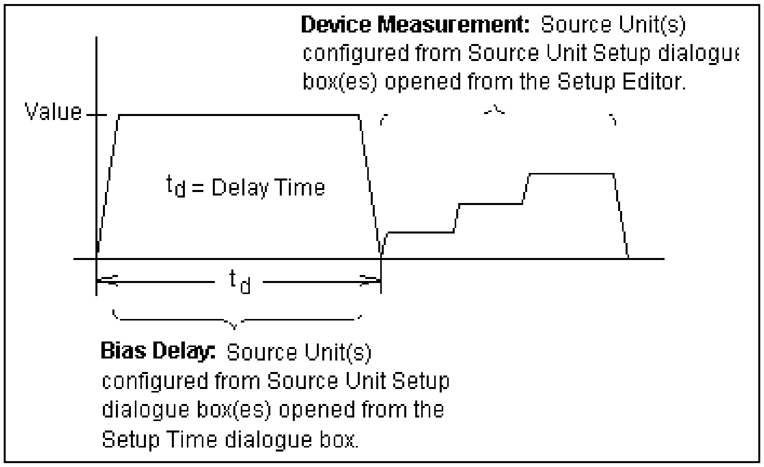
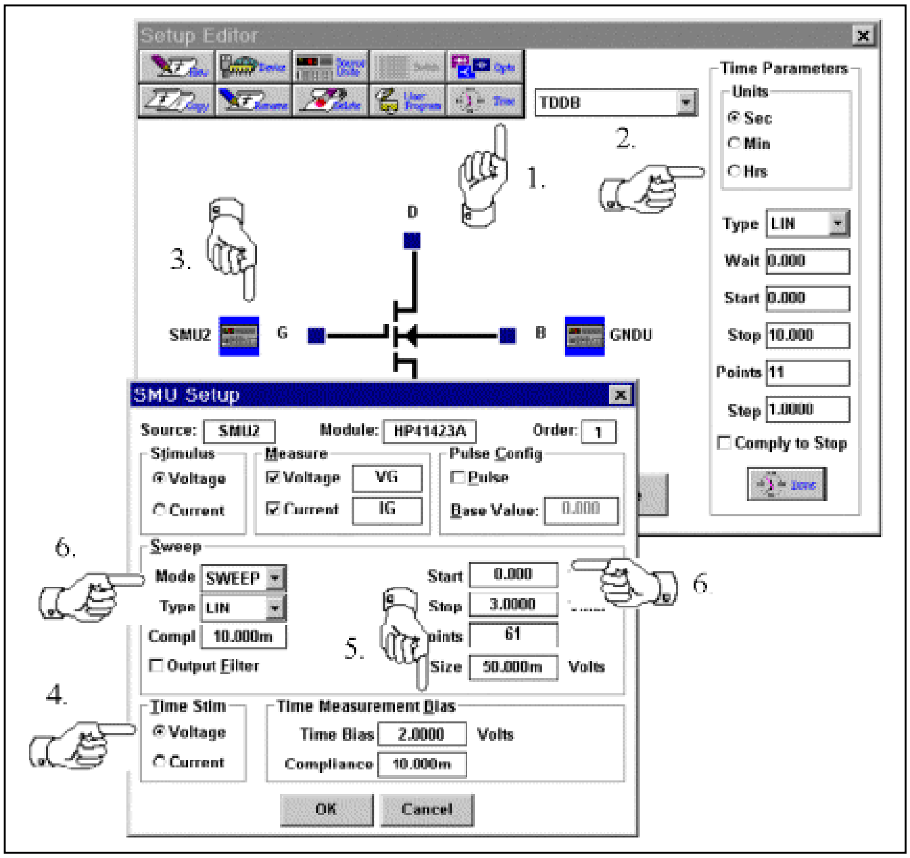


Figure 2-15: An Example of a Source Unit Output Signal Configured in Bias Delay Mode.



How to Setup ICS Bias Delay Measurement Parameters:

1. Display the ICS 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.

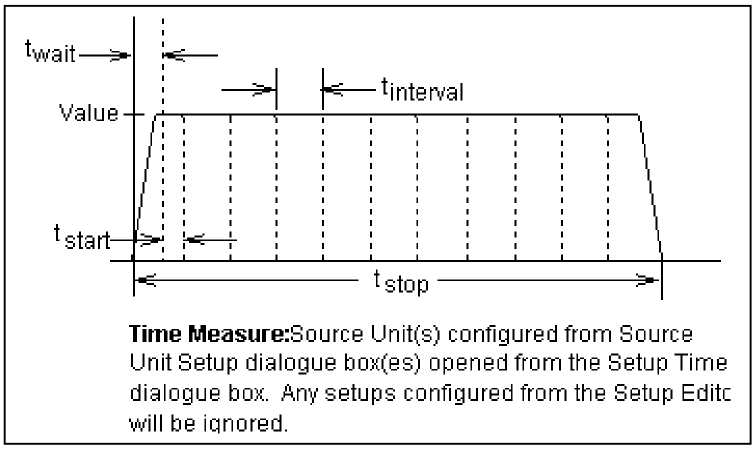
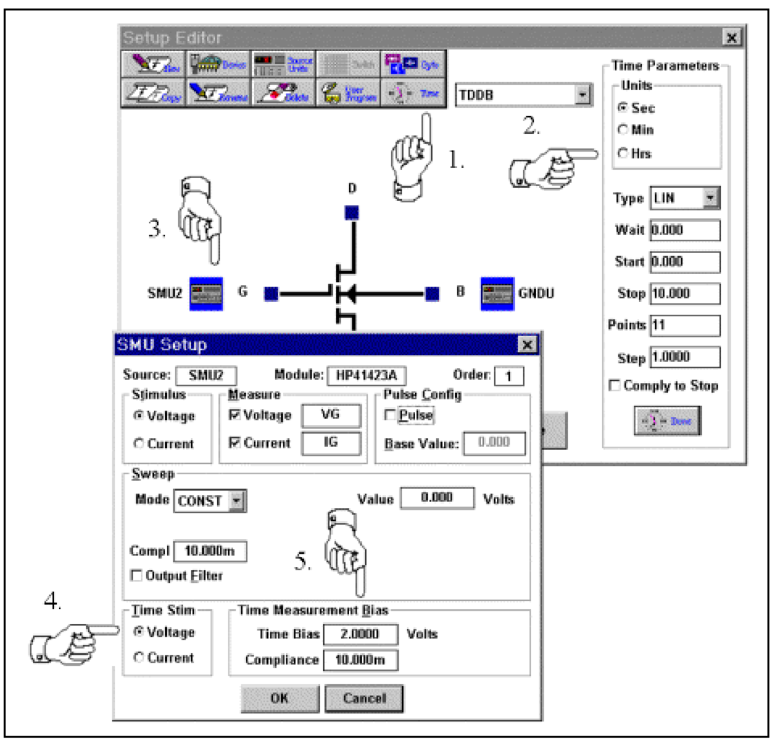


Figure 2-16: An Example of a Source Unit Output Signal Configured in Time Measure Mode.



How to Setup ICS Time Measurement Parameters:

1. Display the ICS 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.

* 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.

Sequence Bias Mode

The Sequence Bias mode allows the user to use two instruments to perform a single measurement. One instrument is used only as a power supply and cannot be used for measuring during the Sequence Bias measurements.

* During the Sequence Bias measurement, each designated source in the Sequence Bias setup will apply a constant signal to the device under test while all of the other measurements in the sequence are performed.

* The Sequence Bias sources are debiased after the completion of the sequence or when the instruments used as the sequence bias source is used for a different type of measurement in the sequence.

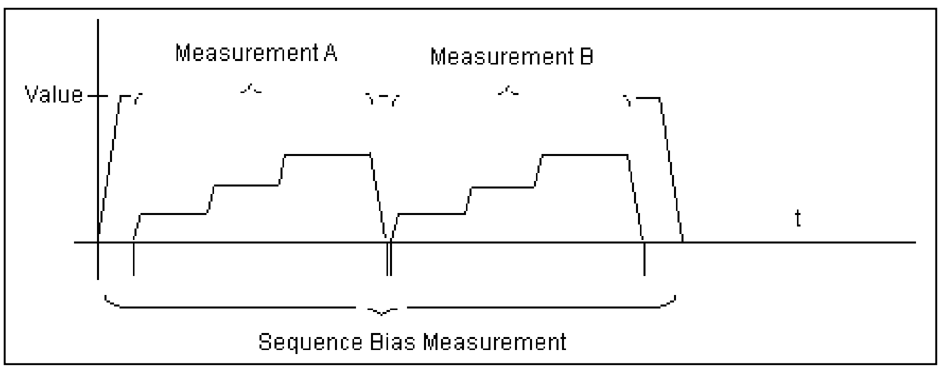


Figure 2-17a: An Example of a Source Unit Output Signal Configured in Sequence Bias Mode.

Sequence Bias Mode Example

Imagine that you have a three-pin structure which you need to make capacitance measurements between pins one and two while holding pin three at constant bias. This is the test that is described below.

Test Setup

1. Create an ICS setup which contains a single constant source for pin three to be used in Sequence Bias Mode as bias only instrumentation. This setup is now referred to as the Bias Setup.
2. Set the Measure remote control to Seq Bias.
3. Create an ICS measurement to perform a capacitance measurement between pin one and pin two.
4. Open the Sequence editor and create a sequence with the Bias Setup followed by the capacitance measurement. (For a complete description of Sequence Measure modes, please refer to *Chapter 4: Measuring Devices*.)

Test Execution

After setting up the above test press the Sequence measurement button to cause the following to occur:

1. Execution of Bias setup. The bias conditions will remain on until either the end of the sequence is reached or until the instrument setup for Bias setup is used in another measurement in the sequence.
2. Execution of CV measurement with the Bias setup as a bias source.
3. After the CV measurement is completed the Bias setup will debias it's instrumentation.

Sequence Stress Mode

The Sequence Stress mode allows the user to stress the device with a constant bias signal before performing multiple device measurements. This mode acts as an extension of the Bias Delay Mode.

* During the Sequence Stress measurement, each designated source unit will apply a constant signal to the device under test. After the delay time, all other device measurements selected in the sequence are performed as defined in the Setup Editor. ICS repeats the execution of the sequence until all of the Stress Sequence stress periods have been performed.

* The Sequence Stress sources are de-biased after the current stress period is completed.

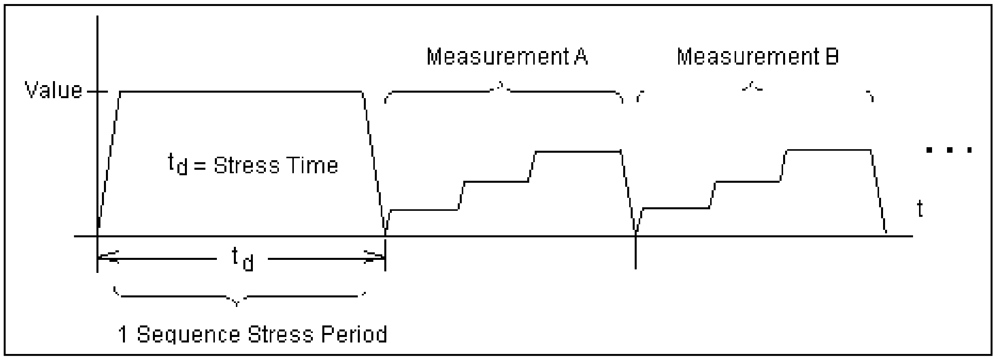


Figure 2-17b: An Example of a Source Unit Output Signal Configured in Sequence Stress Delay Mode.

Sequence Stress Mode Example

Test Setup

1. Create an ICS setup which contains all constant sources to be used in Sequence Stress Mode as stress only instrumentation. This is the same as the example for Bias Delay, thus the bias conditions must be set in the Time Stim area of the SMU setup. This setup will be referred to as the Stress setup.
2. Set the time parameters in the Time Param control for the Sequence Stress measurement to perform the desired number and length of stress periods.
3. Set the Seq Stress mode on the Measure remote control.
4. Create all other ICS setups to perform desired measurements after each stress period. For example Vth and Rds(on).
5. Create a measurement sequence using the ICS sequence editor to execute the Stress setup followed by the Vth and Rds(on) setups. (For a complete description of Sequence Measure modes, please refer to *Chapter 4: Measuring Devices*.)
6. Press the Auto Seq button to execute the measurement.

Test Execution

After setting up the above test press the Sequence measurement button to cause the following to occur:

1. Execution of Stress setup, period 1.
2. Execution of Vth setup.
3. Execution of Rds(on) setup.
4. Execution of Stress setup, period 2.
5. Execution of Vth setup.
6. Execution of Rds(on) setup.
7. Execution of Stress setup, period 3.
8. Execution of Vth setup.
9. Execution of Rds(on) setup.
10. Execution of Stress setup, period 4.
11. Execution of Vth setup.
12. Execution of Rds(on) setup.
13. And so on until all stress periods are completed.

Time Measure Controls

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 de-biasing 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 de-biased 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 ICS should stop the measurement at the stop time even if the number of points or measurements have not been completed.

Optimizing ICS for the Fastest Measurement Interval

Depending upon the instrumentation and GPIB hardware, the GPIB overhead required to obtain each time measure data point could be anywhere in the range of tens to hundreds of milliseconds. Time domain activity is timed with the CPU clock. Because the reporting precision of most CPU clocks is 18.5ms, this value is theoretically the minimum time interval obtainable if no time would be required to actuate the instrument over the IEEE-488 bus. Because this latter point is not true, the range of tens to hundreds of milliseconds is specified as the minimum time interval.

* If you specify a time interval of 100ms or less, ICS will display a warning message acknowledging that data may not be obtainable at the specified interval. ICS will always measure the specified number of points, even if ICS cannot maintain the specified time interval. The resultant TIME values displayed in the time measure data window are ACTUAL times, not specified times.

Despite the limitations inherent to the time measure mode, here are a few things you can do to achieve the shortest time interval possible:

1. Eliminate all disk I/O activity. This is done from the controls in the Workspace dialogue box. The Workspace dialogue box is opened by selecting OPTIONS/WORKSPACE from the measurement mode menu bar:
    A. Turn OFF Save Data to Temp switch.
    B. Turn OFF the Autostore switch.
    C. Turn OFF the Log File switch.
2. To the degree the instrument allows, do not AUTORANGE the stimulus/measure signals. When possible, configure the stimulus/measure controls to operate on a FIXED range.

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. ICS 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.

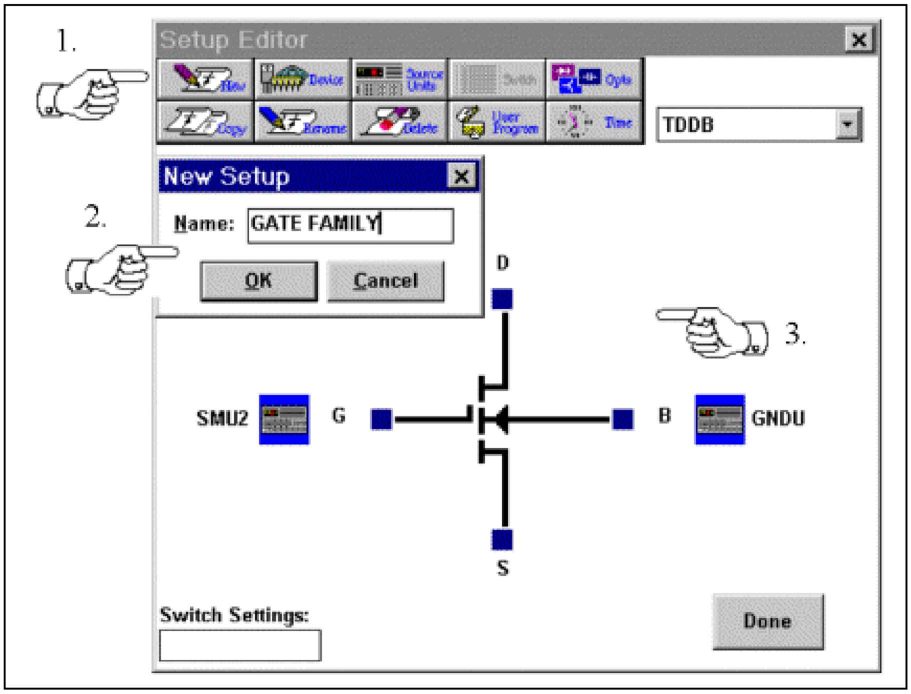


Figure 2-18: How to Create a New Test Setup.

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.

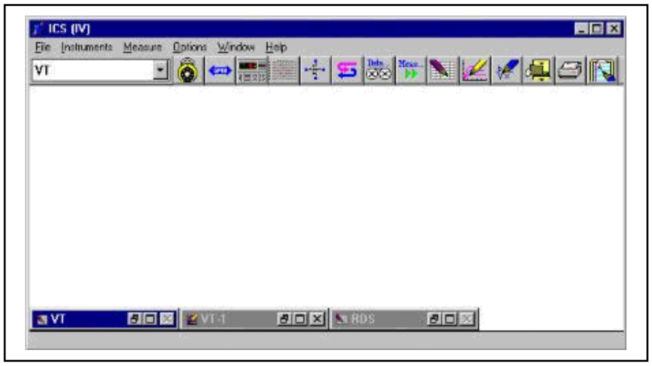


Figure 2-19: Each Data Window Spreadsheet Minimized At the bottom of the desktop corresponds to a Test Setup.

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 2-20: How to Designate a Test Setup from the Tool Bar.

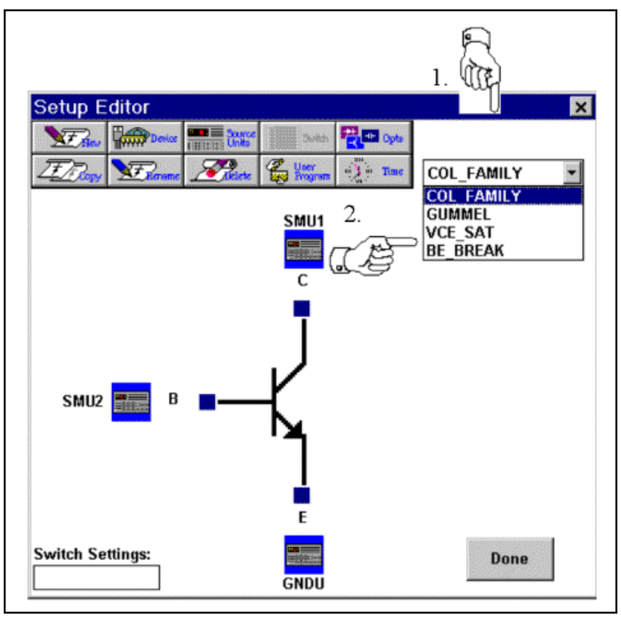


Figure 2-21: How to Designate a Test Setup from the Setup Editor.

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.

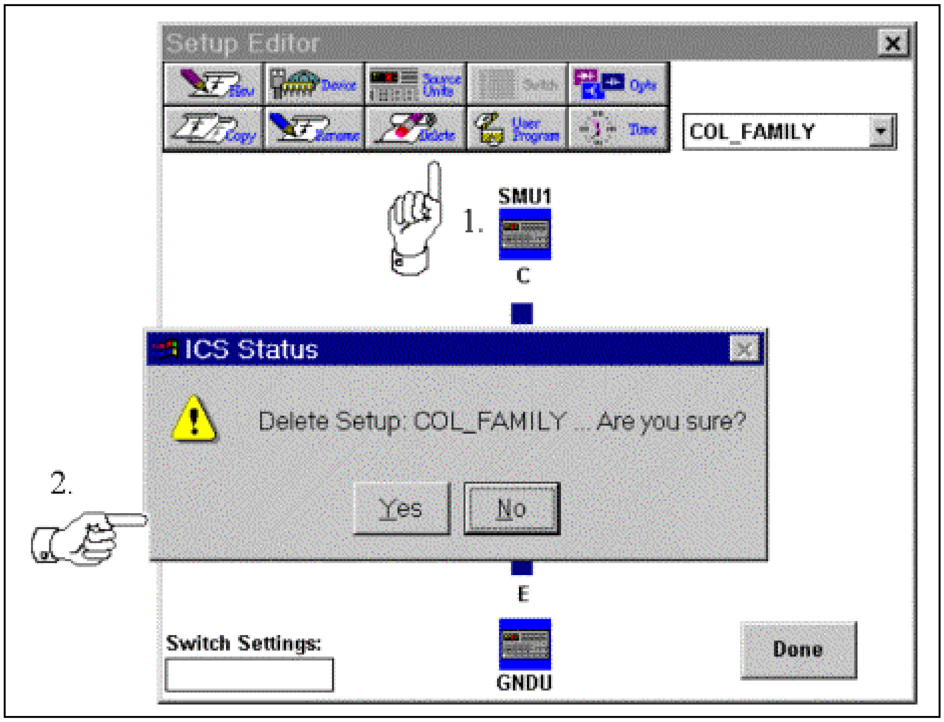


Figure 2-22: How to Delete a Test Setup.

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. ICS 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.

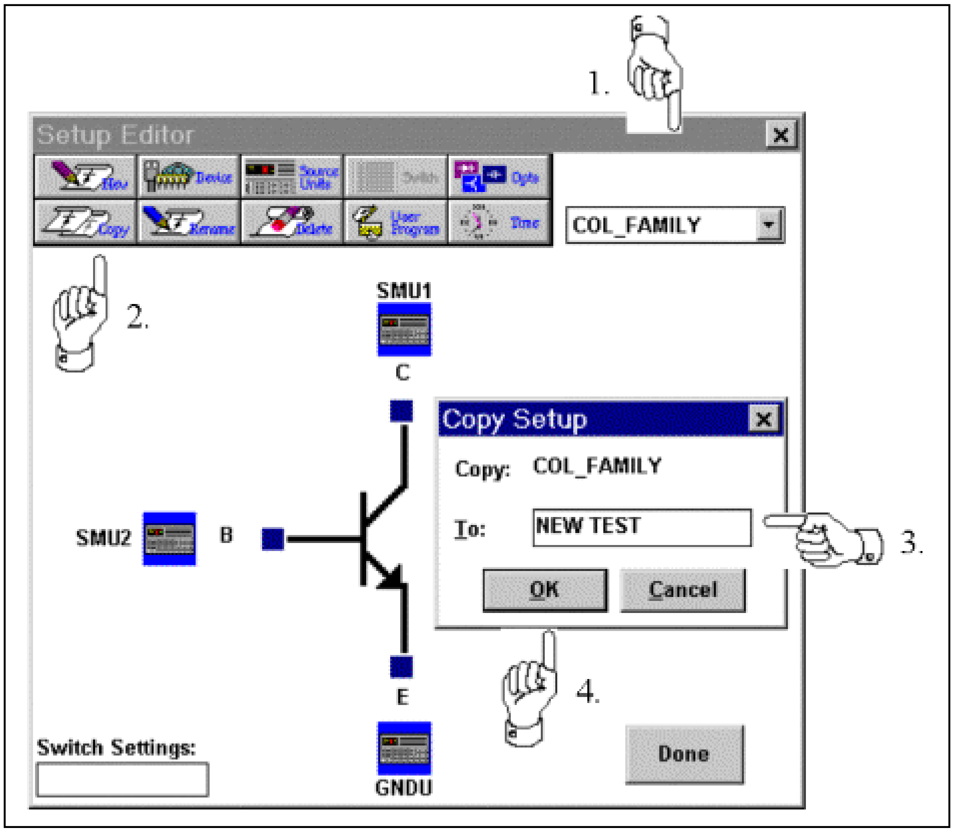


Figure 2-23: How to Copy a Test Setup.

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

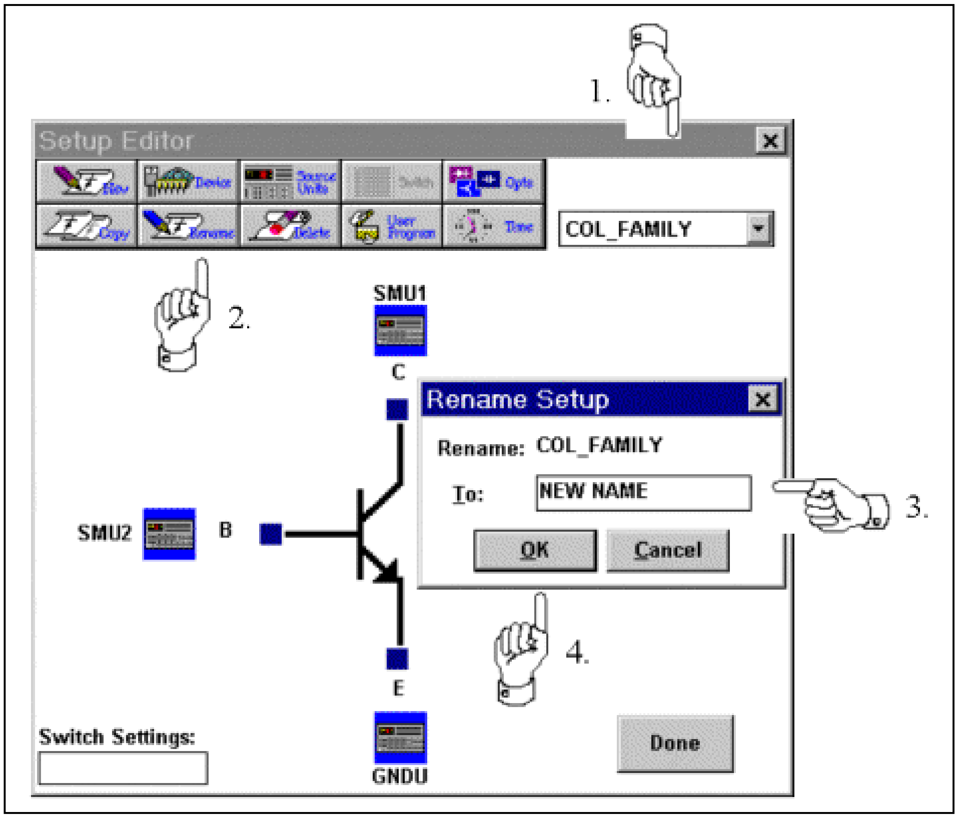


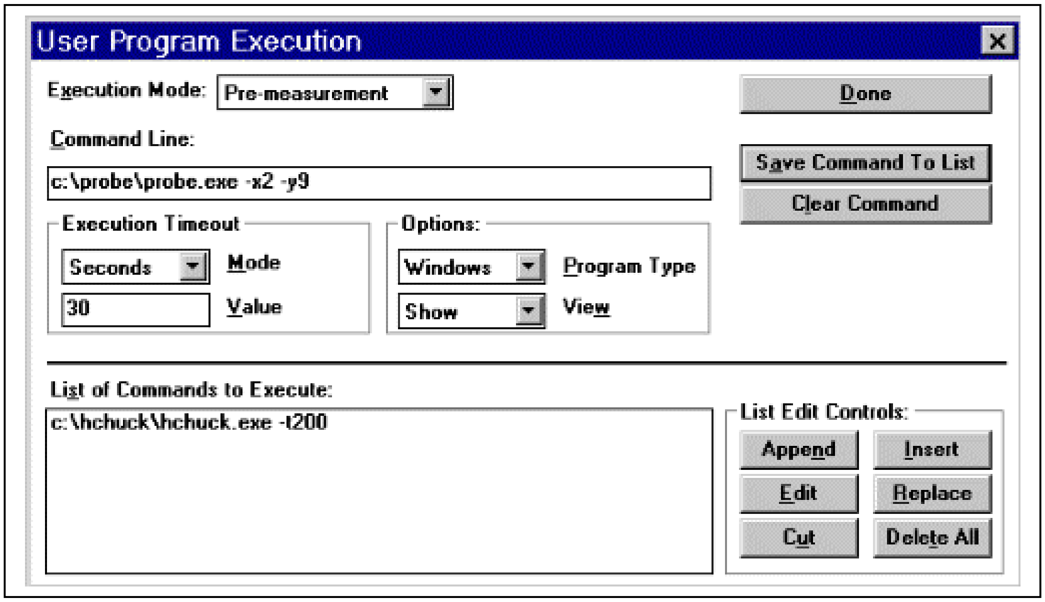
Figure 2-24: How to Rename a Test Setup.

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.



Execution Mode

Sets user/external program to perform before or after the measurement of your test file. Select:
Pre-measurement - Perform user/external program before the measurement.
Post-measurement - Perform user/external program after the measurement

Command Line

Sets command line to be executed to perform user/external 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 ICS 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 with 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.

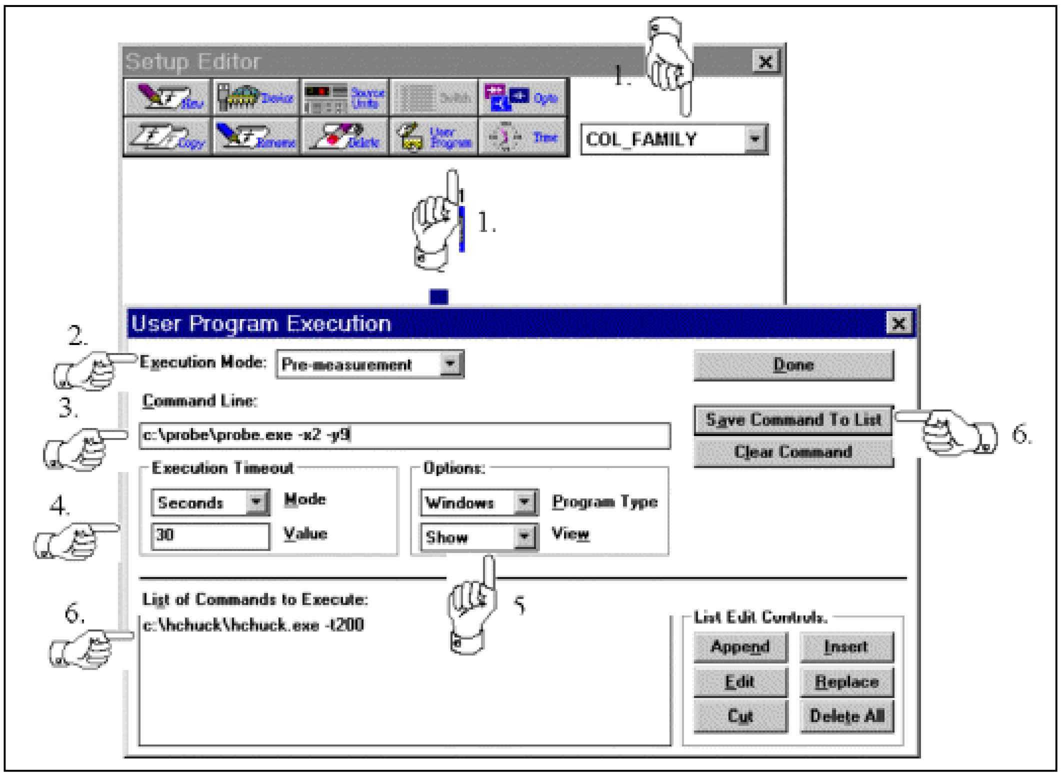


Figure 2-25: How to Define a User Program for Execution.

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.

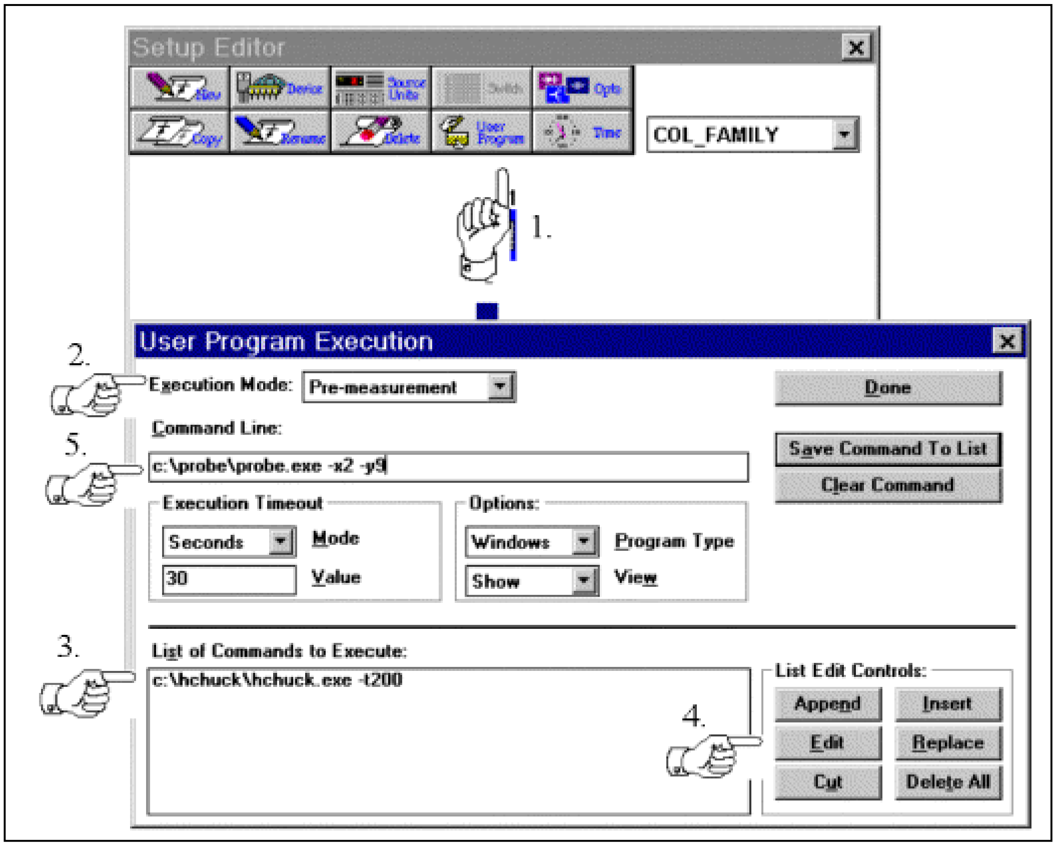


Figure 2-26: How to Edit User Program Execution.

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.

User Program Execution Notes

* Communication with ICS 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 ICS and External Programs

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

Using Switch Settings

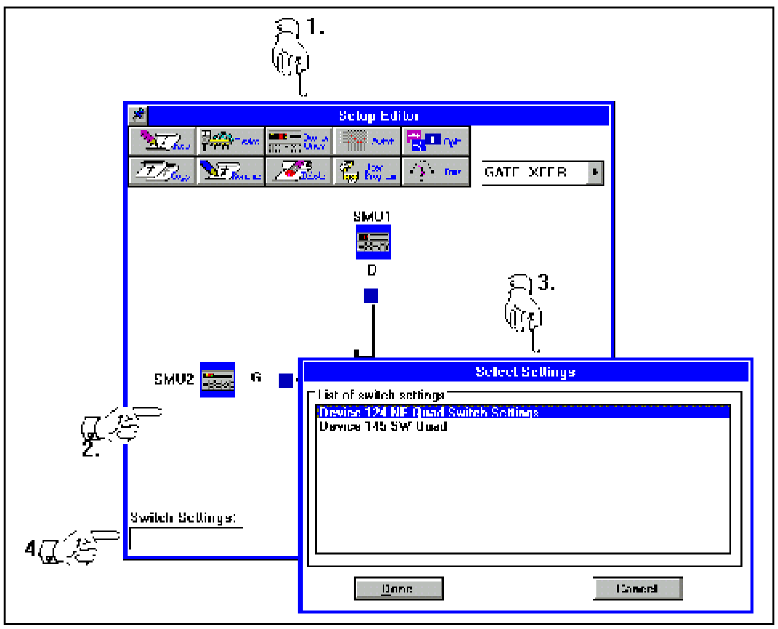


Figure 2-27: How to Select a Switch Setting for Use.

How to Select a Switch Setting for Use:

1. From the Setup Editor setup window, select the SWITCHES button to display the Switch Application.
2. Select the desired switch settings from the list.
3. Press the Done button.
4. The description of the switch settings will be displayed in the Setup Editor's Switch Settings field.

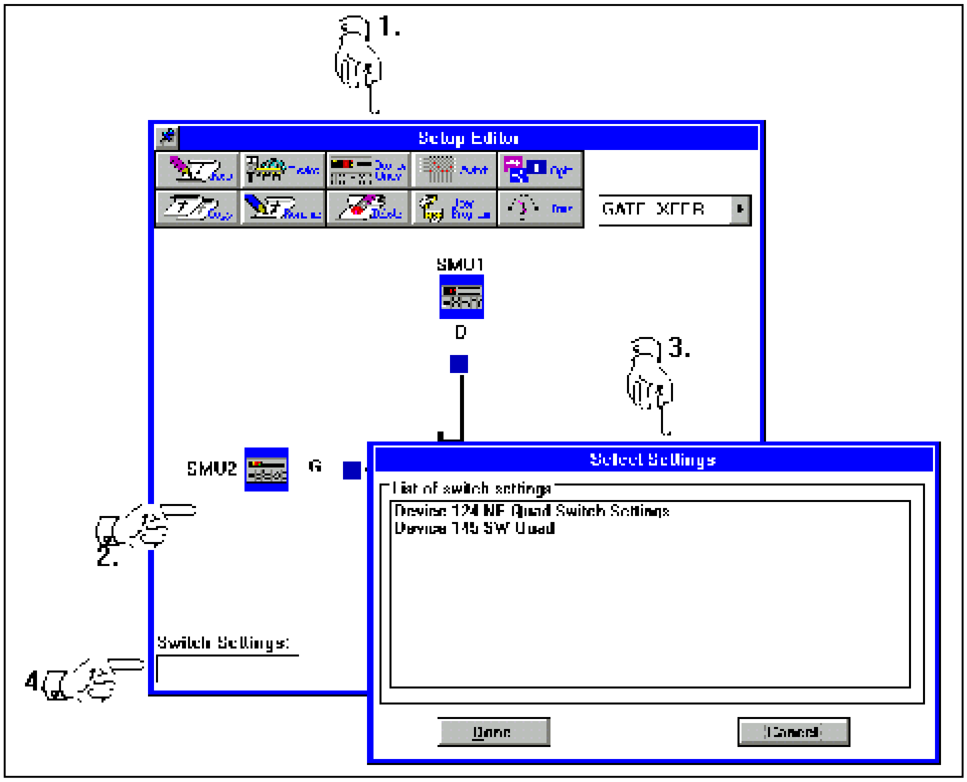


Figure 2-28: How to Remove Switch Settings from Use.

How to Remove Switch Settings from Use:

1. From the Setup Editor setup window, select the SWITCHES button to display the Switch Application.
2. Select no switch settings from the list.
3. Press the Done button.
4. The Setup Editor's Switch Settings field will be cleared designating that no switch settings will be used.

Instrument Options

The final button in the Setup Editor is used to set the instrument specific options available. Please refer to the chapter describing your specific instrument for more information regarding these options.

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.

A project file includes:

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

* When a project file is opened, the necessary instrument driver(s) are automatically connected to ICS. Any of the existing test setups can be executed immediately.

How the File Manager Works

In a conventional DOS environment, files are created by specifying an eight-character name followed by a three-character extension. This naming scheme is limited in sophistication and intuition especially when trying to organize a large quantity of files. When trying to organize hundreds of files, you are forced to resort to some sort of encrypted file naming system. Such a system is often confusing and hard to manage.

* ICS' File Manager eliminates the inconveniences of a DOS environment. The File Manager works in a way that is similar to a combination lock. Instead of identifying a project file by a single filename, project files are identified by specifying up to six project file attributes. Each unique combination of attributes identifies a unique project file. This method is quite versatile because project files can be identified in a way that is more descriptive of the data than conventional DOS naming schemes.

Project file attributes are specified in the File Manager dialogue box. After the attributes are specified, ICS assigns a DOS filename and locates the file in a network of directories and sub directories. This function is entirely transparent to the user. A project file is opened by displaying the File Manager and selecting the attribute designations specified when the project file was saved.

The File Manager Dialogue Box
A project file is defined and opened by specifying the appropriate combination of attribute designations in the File Manager dialogue box. The File Manager dialogue box includes six attribute fields and a COMMENTS field.

* The File Manager attribute fields are hierarchical in nature. This means that the first attribute field must be designated before the second, and the second before the third, the third before the fourth, and so on.

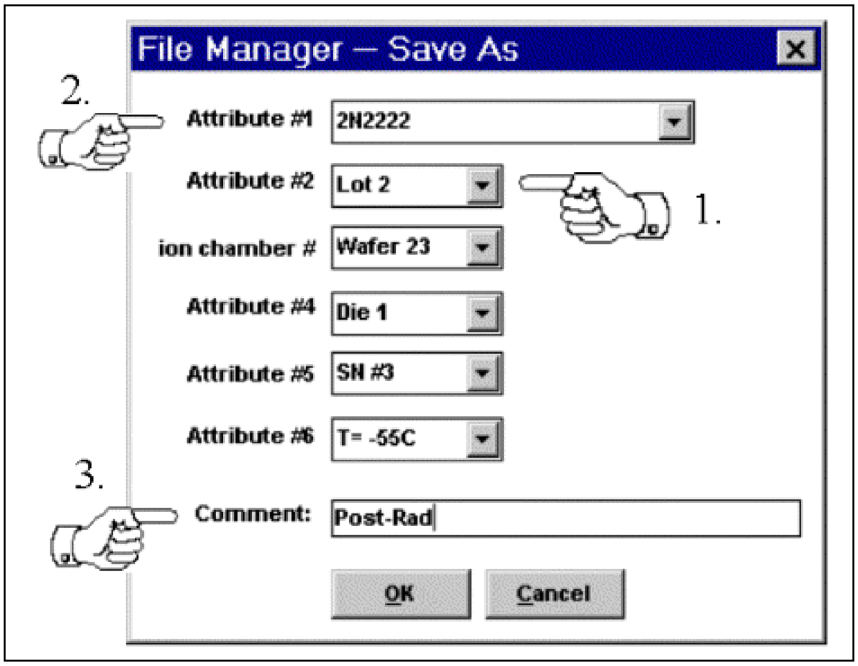


Figure 3-1: The File Manager Dialogue Box.

Attribute Field: Each attribute field is used to designate a characteristic of the project file. A project file is identified by a combination of up to six attribute field designations.
Attribute Field Label: Each attribute field includes a user-defined text label. As a default, the attribute fields are identified as Attribute #1 through Attribute #6. Attribute field labels can be customized using the File Manager Setup dialogue box.
Comment Field: The comment field can be used to specify miscellaneous information. The comment field is not an attribute field; in other words, changing the information in the comment field only is not sufficient to designate a new project file.

The diagram below shows a collection of project files. Notice that the attribute fields are hierarchically related to each other. In other words, the attributes available at any particular level are a subset of the previously designated attribute. Any possible path through the attribute tree identifies a single project file.

* The File Manager was designed with the intention of using the attribute hierarchy to designate increasingly specific details about the project file. With this in mind, the first attribute field is intended to identify a family of related project files. The designation in the ATTRIBUTE #1 field should relate to some characteristic that is common to each project file in the family, such as device type. Each additional attribute level is used to designate a more specific group of related project files. A single project file is uniquely identified by specifying the last attribute.

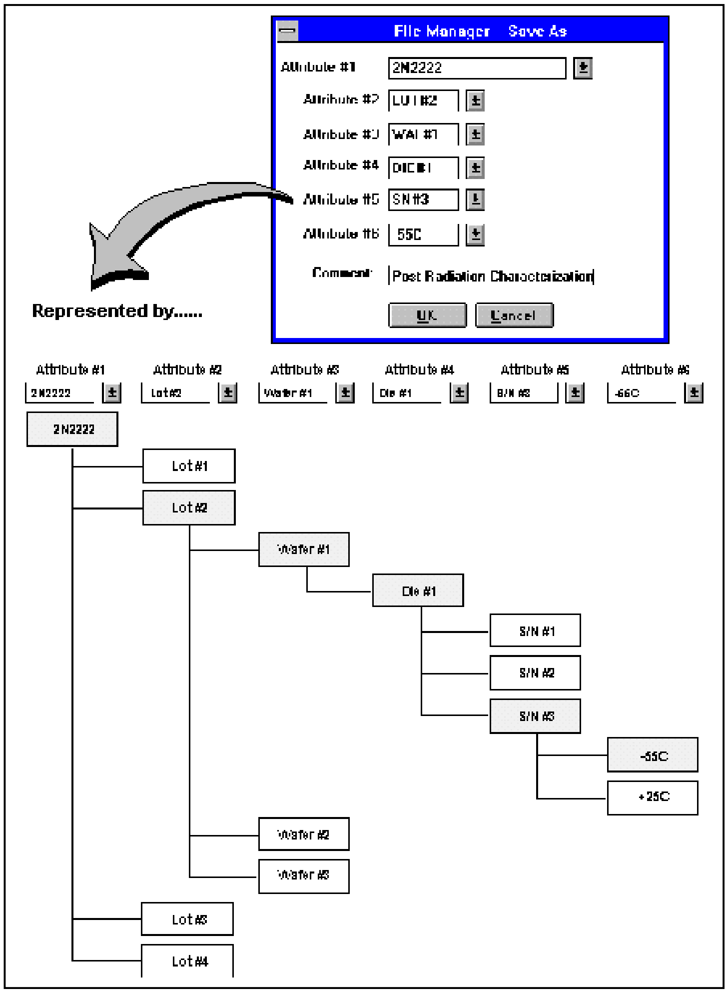


Figure 3-2: A Project File is Identified by Specifying up to 6 Project File Attributes.

Storing Project Files

* The project file storage mode designates the maximum number of attribute fields available in the File Manager. The File Manager can be toggled between Package Mode and Wafer Mode.

Package Mode allows the user to save project files by designating up to three attributes, while Wafer Mode allows the user to designate up to six attributes. When configured in Package Mode, only the ATTRIBUTE #1, ATTRIBUTE #5, and ATTRIBUTE #6 fields are available. When configured in Wafer Mode, all six attribute fields are available. The criteria for determining the project file storage mode is the desired level of detail with which you wish to identify project files.

Wafer Mode

ICS is shipped with the Wafer Mode selected. In some cases, you may want to identify project files using a hierarchy of attributes six levels deep. An example of such an application is the need to store data as a function of device name, lot, wafer, die, serial number, and temperature. A small collection of project files created with the File Manager set to Wafer Mode is shown on the following page. Notice that project files are identified using all six of the available attribute fields.

* When defining a project file in Wafer Mode, it is not necessary to specify a designation in all six attribute fields. Project files can be created with as little as an ATTRIBUTE #1 designation. However, when designating multiple attributes, each attribute must be designated in consecutive order due to the hierarchical implementation of the File Manager.

The default attribute labels in Wafer Mode are ATTRIBUTE #1 through ATTRIBUTE #6. All six attribute labels can be customized by the user. Refer to *File Manager Attribute Labels* later in this chapter.

* Project files saved with the File Manager set to Wafer Mode must be opened in Wafer Mode. Any project files saved in Wafer Mode will not be accessible when the File Manager is set to Package Mode.

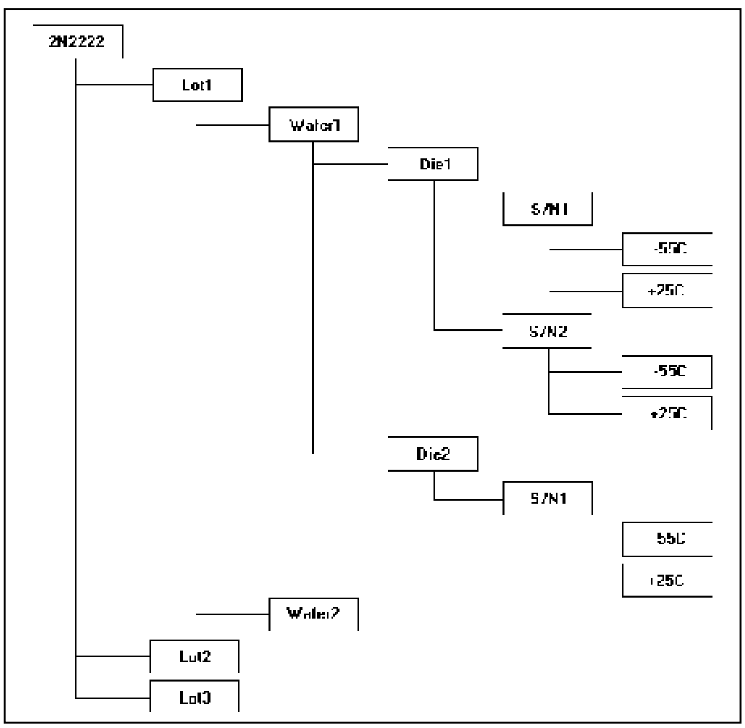


Figure 3-3: A Collection of Project Files Created with the File Manager Set to Wafer Mode.

Package Mode

* Project files can be designated using a maximum of three attributes by configuring the File Manager in Package Mode.

Project files can be saved in Wafer Mode using only three attribute fields, but not with the same three attribute fields accessible in Package Mode. Project attributes must be specified in consecutive order due to the hierarchical nature of the file management system. If you want to create a project file in Wafer Mode using only three attributes, you must specify a designation in the ATTRIBUTE #1, ATTRIBUTE #2, and ATTRIBUTE #3 fields. In Package Mode, three of the six attribute fields are unavailable. A project file is defined by specifying a designation in the ATTRIBUTE #1, ATTRIBUTE #5, and ATTRIBUTE #6 fields. This capability is convenient because the ATTRIBUTE #5 field is the field that is automatically sequenced when running ICS in Autostore and/or LogFile mode.

* The available attribute fields in Package Mode are the ATTRIBUTE #1, ATTRIBUTE #5, and ATTRIBUTE #6 fields. The unavailability of ATTRIBUTE #2 through ATTRIBUTE #4 is indicated by the gray color of the field labels. Even though there are three inactive attribute fields, all six attribute field labels can be customized by the user. Refer to *File Manager Attribute Labels* later in this chapter.

* Project files saved with the File Manager set to Package Mode must be opened in Package Mode. Any project files saved in Package Mode will not be accessible when the File Manager is set to Wafer Mode.

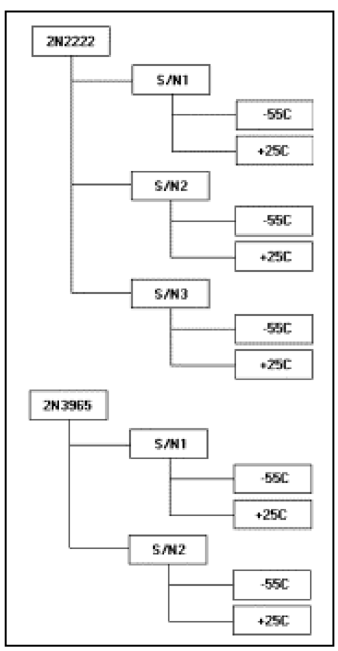


Figure 3-4: A Collection of Project Files Created with the File Manager Set to Package Mode.

Viewing the Database from DOS

* The database directory designated in the File Manager Setup dialogue box is called the database root directory (see *Setting the File Manager Options*). If the default installation path is accepted during the execution of the ICS Setup Program, the database root directory will be C:\ICS\DATA.

Project files are stored in a cascade of sub directories located under the database root. This sub directory structure is created at each ICS database location.
The database sub directory structure is shown below. ICS stores the first project file in the lowest sub directory and works upward as more project files are added. The sub directory cascade allows the user to store literally millions of project files.

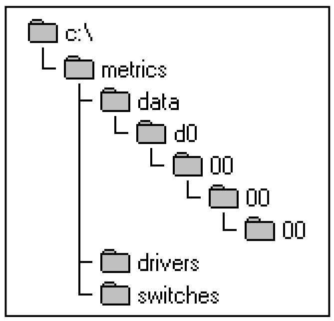


Figure 3-5: ICS' Database Sub directory Structure.

All project files are named "RESULTS" followed by a sequence number. The first project file is named "RESULTS.00" and is stored in the last "00" sub directory. The second project file is written to the same sub directory but is labeled "RESULTS.01". The third is labeled "RESULTS.02", and so on. A single sub directory will hold 100 project files.

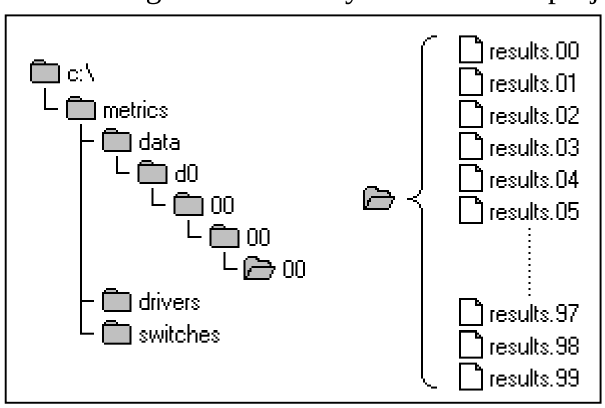


Figure 3-6: Each Project File is Named "RESULTS" Followed by a Sequence Number. A Single Sub directory can Hold 100 Project Files.

After "RESULTS.99" is written to the last "00" sub directory, a new sub directory named "01" will be created parallel with the "00" sub directory just filled. The first project file written to the new "01" sub directory will be labeled "RESULTS.00". After "RESULTS.99" is written to the "01" directory, a new sub directory named "02" will be created parallel with the "01" and "00" directories. This process continues until 99 parallel sub directories are created at the bottom of the cascade. When sub directory "99" is full, a new sub directory named "01" will be created parallel with the second-to-last "00" sub directory. This process continues all the way through the "D0" sub directory at the top of the cascade. Using this file storage algorithm, a single database location can store ten million project files.

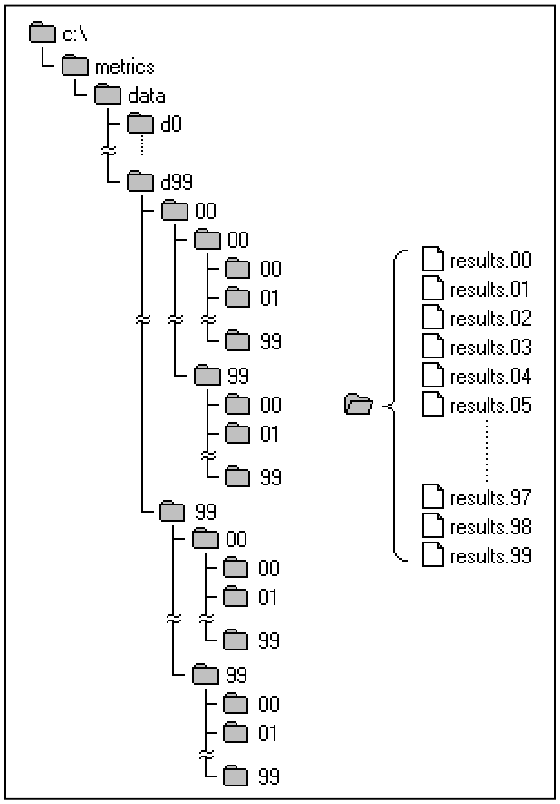


Figure 3-7: ICS' File Management System can Manage up to 10^6 Project Files.

* The operation of the sub directory cascade is completely transparent to the user. The user creates and identifies project files by specifying a combination of up to six attributes in the File Manager dialogue box.

When a project file is saved for the first time, ICS reads the specified attributes in the File Manager and calculates a numerical string that "points" to the corresponding "RESULTS.XX" file and the directory location. These "pointers" are never seen by the user but are necessary to correlate the File Manager attribute specifications with the proper project file. The project file pointers are stored in the DEVTYPE.KEY file located in the database root directory.

WARNING! Never copy or move project files (RESULTS.XX) from a DOS-environment! Doing so will scramble the correlations between the File Manager attribute configurations and the project files. All project file manipulations MUST be performed from ICS in order to preserve the integrity of the database!

Setting the File Manager Options

The File Manager Setup dialogue box allows the user to customize certain features of the File Manager and the File Manager dialogue box. The following features can be controlled by the user:

1. **Project File Storage Mode:** The Project File Storage mode can be toggled between Wafer Mode and Package Mode. In Wafer Mode, the File Manager functions with a six-layer attribute hierarchy. All six attribute fields are available. In Package Mode, the File Manager functions with a three-layer attribute hierarchy.
2. **Database Root Directory:** In order to eliminate the size restrictions that may be encountered when creating a database on your hard driver, you can designate the driver and directory location of the ICS database(s). Database(s) can be located on any local or network drive. This allows you to install ICS locally, but store the data on a Local Area Network (LAN).
3. **Attribute Field Labels:** ICS allows the user to customize the text labels at the base of the File Manager attribute buttons.

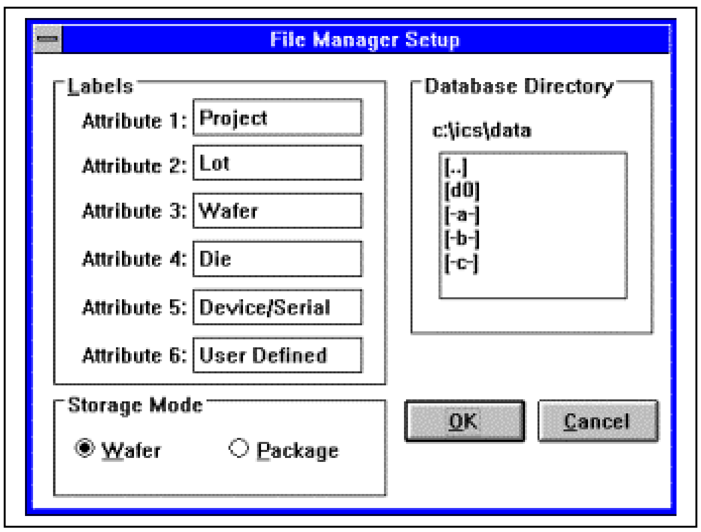


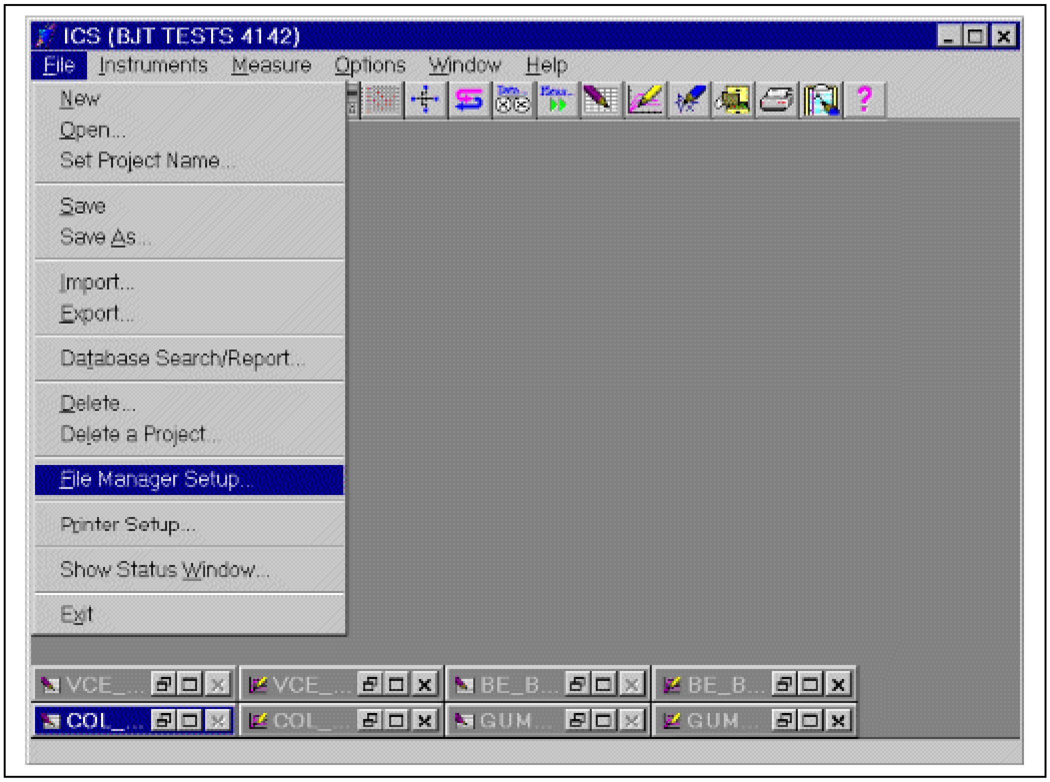
Figure 3-8: The File Manager Setup Dialogue is used to Customize Certain Features of the File Manager and the File Manager Dialogue Box.

How to Display the File Manager Setup Dialogue Box:

1. Select FILE/FILE MANAGER SETUP from the menu bar.

Alternatively,

1. Choose the OPTIONS/WORKSPACE selection from the measurement mode menu bar. This will open the Workspace dialogue box.
2. From the Workspace dialogue box, click the File Manager Setup button.



OR

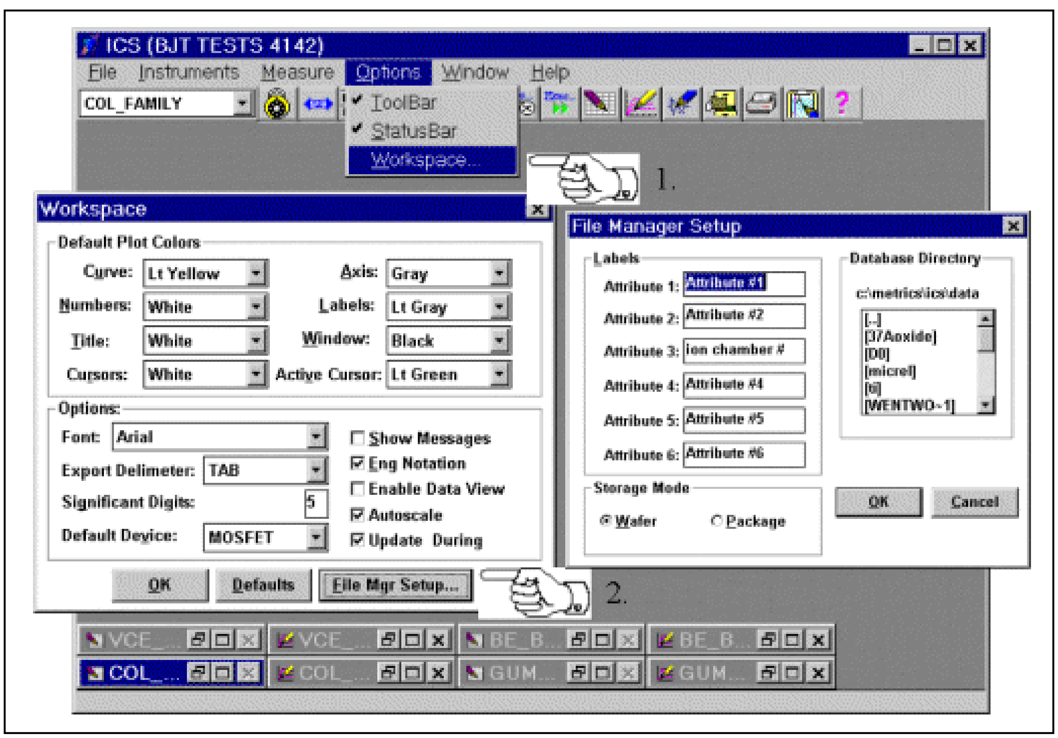
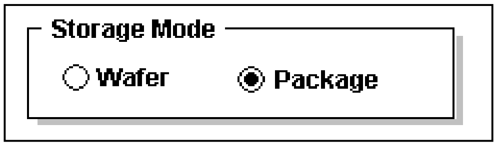


Figure 3-9: How to Display the File Manager Setup Dialogue Box.

Designating the Project File Storage Mode



The project file storage mode designates the maximum number of attribute fields available in the File Manager.

The complexity of the file manager can be toggled between Package Mode and Wafer Mode. Package Mode allows the user to identify project files using a maximum of three attributes, while Wafer Mode allows the user to identify project files using a maximum of six attributes. When configured in Package Mode, only the ATTRIBUTE #1, ATTRIBUTE #5, and ATTRIBUTE #6 fields are available. When configured in Wafer Mode, all six attribute fields are available.

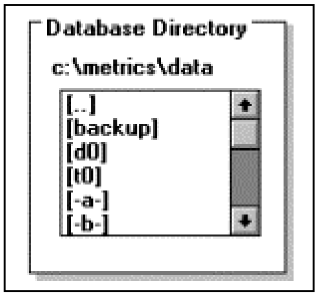
To designate the desired project file storage mode, select the appropriate switch in the File Manager Setup dialogue box. For more information about the project file storage mode, see *Storing Project Files* earlier in this chapter.

The project file storage mode can be toggled from one mode to the other while a project file saved under the original format is open. For example, if the project file storage mode is presently set to "Wafer", the storage mode can be toggled to "Package" even though a "Wafer" project file is loaded. If FILE/SAVE is selected, the project file will be saved according the storage format of the file regardless of which mode is active. If FILE/SAVE AS is selected, the project file will be saved according to the active storage mode. When a project file is saved with a new storage format using FILE/SAVE AS, the original project file (designated according to the other mode) is still preserved. This arrangement allows the user to copy project files from one storage mode to the other.

How to Copy a Project File from one Storage Mode to the Other:

1. In the File Manager Setup dialogue box, designate the storage mode that corresponds to the present storage mode of the project file you wish to copy.
2. Open the desired project file as described in *Opening a Project File* later in this chapter.
3. In the File Manager Setup dialogue box, designate the alternative storage mode.
4. From the toolbar or the measurement mode menu bar, select FILE/SAVE AS. This will open the File Manager dialogue box. Notice that the availability of the attribute fields will correspond to the storage mode designated in Step #3, not the storage mode of the project file.
5. Designate the relevant attributes as desired. If necessary, refer to *Saving a Project File* later in this chapter. Click the File Manager OK button.
6. The original version of the project file is still preserved. Delete the original project file if desired.

Specifying the Database Root Directory



While the file manager stores project files in a combination of directories and sub-directories, the user can specify the drive location and root directory of the database. ICS allows you to designate any drive and directory (local or network) as a location for data storage. At any time you can designate an alternative database by selecting a new drive location and root directory. This feature provides the user with the ability to operate among multiple databases.

As a default, the ICS Setup Program will create a single database directory under the ICS root directory. If the default installation drive is accepted while running the ICS Setup Program, the database root directory will default to C:\ICS\DATA.

* You cannot create a database directory from ICS. You CAN designate an existing directory as the database location.

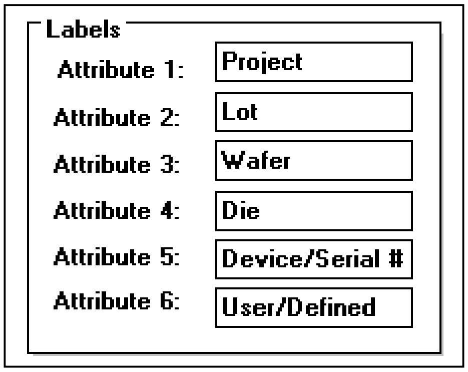
To create a new database directory, first create the directory from a DOS-window (minimize ICS and double-click on the DOS-prompt; type "EXIT" when done). Open the File Manager Setup dialogue box as outlined earlier in this chapter. Select the new directory from the Database Directory window.

Any directory or drive can be designated as the database root directory. For example, it may be advantageous to run ICS locally but locate the database on a LAN, if available. Designating a LAN directory may eliminate the disk space limitation that could be encountered if all of your data is stored on a hard drive. Designating a LAN directory may also be advantageous if your LAN drives are regularly backed-up.

A powerful capability provided by ICS is the fact that you can designate more than one database location. You could locate one database on your hard drive and another database on a LAN drive. There's no limit imposed by ICS to the number of databases you can create. The only practical limit is the disk space availability of your machine or network.

* While you can define as many databases as you like, you can only function in one database at a time. Project files created in one database are not available when functioning in a different database location.

Customizing the Attribute Field Labels



Each of the File Manager attribute field labels can be customized by the user. The default labels are ATTRIBUTE #1 through ATTRIBUTE #6. When configured in Package Mode, the default labels remain the same even though three of the attribute fields are unavailable. However, all six labels can still be customized regardless of the field's status. For example, the user may wish to designate the three inactive fields as "N/A" when configured in Package Mode.

To customize the File Manager attribute labels, open the File Manager Setup dialogue box as outlined earlier in this chapter. The attribute labels are specified in a set of fields identified as "Attribute 1" through "Attribute 6". Edit the field labels as desired.

Saving a Project File

All of the information contained in a project file is automatically written to a temporary file as soon as any specifications are configured or data measurements are completed. The temporary file is stored as RESULTS.DAT in the root directory of the database. If the default specifications are accepted during the installation of ICS, the RESULTS.DAT file will be stored in C:\ICS\DATA. When FILE/SAVE AS is selected, all of the information from the temporary file is copied to the designated project file. After a project file is created, further changes are added to the temporary file. To update the project file, select FILE/SAVE. For more information about the functionality of the temporary file, see *Understanding the Temporary File* later in this chapter.

File/Save As: Creating a New Project File

In a conventional DOS environment, files are created by specifying an eight-character name followed by a three-character extension. This naming scheme is limited in sophistication and intuition especially when trying to organize a large quantity of files. When trying to organize hundreds of files, you are forced to resort to some sort of encrypted file naming system. Such a system is often confusing and hard to manage.

* ICS' File Manager eliminates the inconveniences of a DOS environment. The File Manager works in a way that is similar to a combination lock. Instead of identifying a project file by a single filename, project files are identified by specifying up to six project file attributes. Each unique combination of attributes identifies a unique project file. This method is quite versatile because project files can be identified in a way that is more descriptive of the data than conventional DOS naming schemes.

A project file is created and later identified by specifying the appropriate combination of attribute designations in the File Manager dialogue box. The File Manager dialogue box includes six attribute fields and a COMMENTS field. The File Manager attribute fields are hierarchical in nature. This means that the first attribute field must be designated before the second, and the second before the third, the third before the fourth, and so on.

The diagram on the following page shows a collection of project files. Notice that the attribute fields are hierarchically related to each other. In other words, the attributes available at any particular level are a subset of the previously designated attribute. Any possible path through the attribute tree identifies a single project file.

The File Manager was designed with the intent of using the attribute hierarchy to designate increasingly specific details about the project file. With this in mind, the ATTRIBUTE #1 field is intended to identify a family of related project files. The designation in the ATTRIBUTE #1 field should relate to some characteristic that is common to each project file in the family, such as device type. Each additional attribute level is used to designate a more specific group of related project files until at last a single project file is identified by specifying the final attribute.

The related project files associated with "Wafer1" are displayed in the project file tree on the following page. The family of project files may also include other lot and wafer designations, but they are not shown in this diagram. The gray boxes correspond to the project file attributes designated in the File Manager.

The project file storage mode designates the maximum number of attribute fields available in the File Manager. The File Manager can be toggled between Package Mode and Wafer Mode. Package Mode allows the user to save project files by designating up to three attributes, while Wafer Mode allows the user to designate up to six attributes. For more information about the project file storage mode, refer to *Storing Project Files* earlier in this chapter.

* When saving a project file for the first time, specify a designation in the ATTRIBUTE #1 field, such as the device name or family. The ATTRIBUTE #1 designation can be any alpha-numeric string up to 23 characters in length. Hit the TAB key to advance from one attribute field to the next. Specify increasingly specific details in each consecutive attribute field until all the fields are designated, or until you have designated the number of fields intended. Any alpha-numeric designation is acceptable as long as it fits in the field.

* It is not necessary to specify a designation in all of the available attribute fields. Project files can be defined with as little as a ATTRIBUTE #1 designation. However, when designating multiple attributes, each attribute must be designated in consecutive order due to the hierarchical implementation of the File Manager.

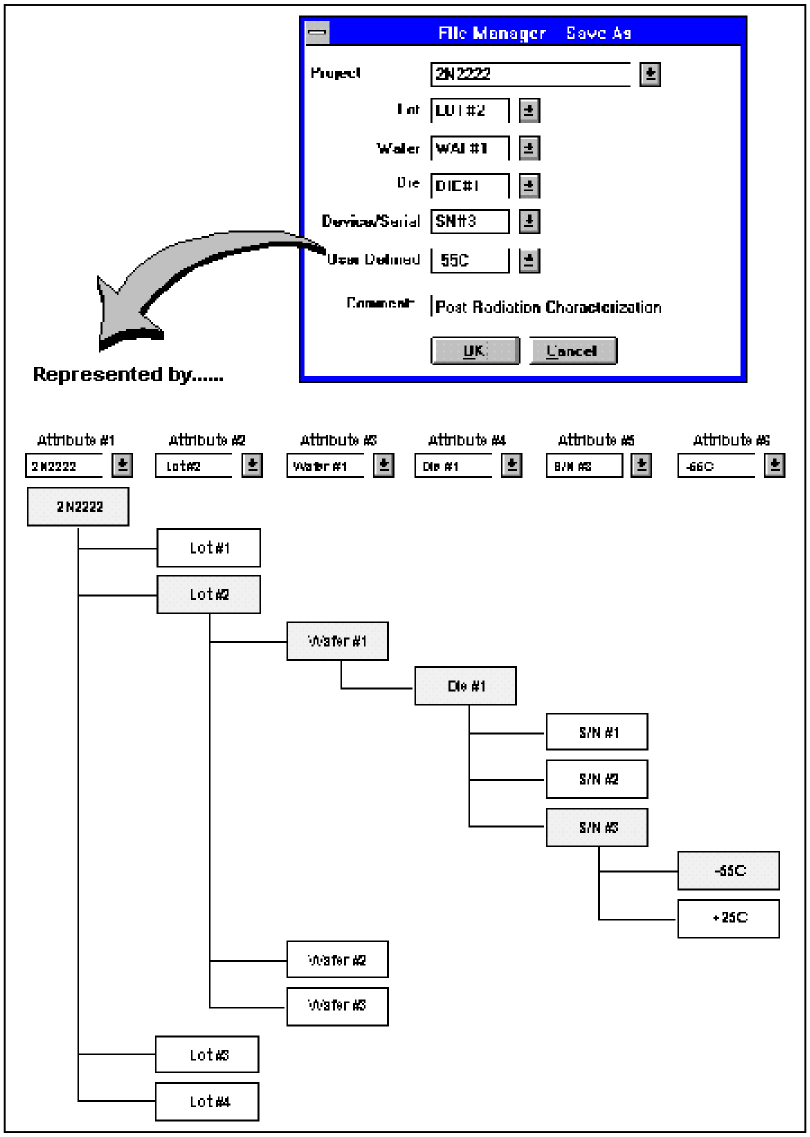


Figure 3-10: A Project File is Identified by Specifying up to 6 Project File Attributes.

How to Create a New Project File:

1. Select the tool bar Save As button, or select FILE/SAVE AS from the measurement mode menu bar. This will open the File Manager dialogue box.
2. The File Manager attribute fields are hierarchically structured. This means that the contents of each previous field must be specified before specifying the contents of the successive fields. Specify the contents of the first attribute field. The contents of the

ATTRIBUTE #1 field can be specified by selecting an existing designation, if available. Click the attribute field scroll arrow to display a list of previously defined attributes. Click on the desired selection.

3. Depending upon whether the File Manager is configured in Package Mode or Wafer Mode, three of the six attribute fields may not be available. If necessary, refer to *Storing Project Files* earlier in this chapter.
4. Specify the contents of each successive attribute field. A list of previously defined attributes can be displayed by clicking on the attribute field scroll arrow. Click on the desired selection or specify a new attribute.
5. At least one of the attribute fields must include a unique designation; otherwise, an existing project file will be overwritten. If this occurs the user will be prompted with a message before the original file is overwritten. If the user selected existing designations in the first five attribute fields, the ATTRIBUTE #6 attribute field must be uniquely specified in order to define a new project file.
6. The Comments field is not an attribute field. Specifying unique contents in the Comments field only is not sufficient to create a new project file.
7. After all of the relevant attribute fields have been specified or designated, select the OK button. Do not use the keyboard return key anywhere in the File Manager. Doing so will close the File Manager and void any specifications or changes just made.

Each time a test setup is defined or data is measured (prior to defining a project file), the corresponding information is recorded in a temporary file. The temporary file is named RESULTS.DAT and is located at the root of the designated database directory. Selecting FILE/SAVE AS copies the contents of the temporary file to the project file identified by the attributes specified in the File Manager.

* If FILE/SAVE AS is selected without defining any project file attributes, the contents of the temporary file will be written to the default project file. The default project file is identified by the designation "NONAME" in the ATTRIBUTE #1 field. No other attributes are defined.

File/Save: Updating the Current Project File

* If File/Save is selected from the menu bar, the active project file will be updated with the most recently measured data and any added test setup(s). Selecting File/Save does not create a new project file. If any test setups have been remeasured, the original data in the project file will be overwritten with the data just measured as soon as FILE/SAVE is selected. If you wish to preserve the data in the original project file, save the most recently measured data as a unique project file by selecting FILE/SAVE AS.

Even though the latest data and any added test setups appear along the desktop, a project file will not be overwritten with the new information until the project file is updated by selecting FILE/SAVE. Each time data is measured or a new test setup is defined (prior to updating the project file), the corresponding information is recorded in a temporary file. The temporary file is named RESULTS.DAT and is located at the root of the designated database directory. Selecting FILE/SAVE updates the open project file with the contents of the temporary file.

If FILE/SAVE is selected without first opening or creating a project file, the contents of the temporary file will be written to the default project file. The default project file is identified by the designation "NONAME" in the ATTRIBUTE #1 field. No other attributes are defined.

Opening a Project File

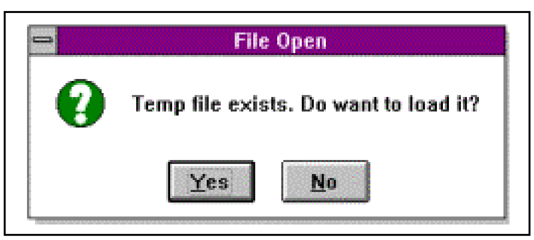
Only one project file can be opened at a time. Project files are opened by designating the same combination of file attributes specified when the project file was saved. Selecting an attribute designation will display any corresponding attribute designations in the next attribute field. For example, selecting a single ATTRIBUTE #1 designation may display three designations in the ATTRIBUTE #2 field. Selecting a single ATTRIBUTE #2 designation may display four designations in the ATTRIBUTE #3 field. This process may continue up to the ATTRIBUTE #6 field. The project file is fully identified when a designation is selected in the ATTRIBUTE #6 field, or there are no designations available in the next attribute field. Selecting OK in the File Manager will open the project file and display the corresponding test setups and data at the bottom of the desktop.

* Only one project file can be opened at a time.

How to Open a Project File:

1. Select FILE/OPEN from the measurement mode menu bar. This will open the File Manager dialogue box.
2. Select the relevant alpha-numeric attribute designations. Click the attribute field scroll arrow to display a list of available options. Click the desired selection.
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.

Loading the Temporary File



If ICS is terminated before having the opportunity to create or update a project file, the user can restore the previous desktop configuration by loading the temporary file.

When the user selects FILE/OPEN, ICS will prompt the user with a message box providing the opportunity to load the temporary file. If you answer YES, the configuration of the desktop will be restored to its configuration just before the application was terminated. After loading the temporary file, store the information permanently by selecting FILE/SAVE AS or FILE/SAVE. If you answer NO, the information in the temporary file will be lost. For a detailed overview of the temporary file and how it works, refer to *Understanding the Temporary File* later in this chapter.

Copying a Project File to a New Database

* A project file can be copied from one database to another. This procedure is limited to one project file at a time.

WARNING! Never copy or move project files (RESULTS.XX) from a DOS-environment! Doing so will scramble the correlations between the File Manager attribute configurations and the project files. All project file manipulations MUST be performed from ICS in order to preserve the integrity of the database!

Project files (RESULTS.XX) must never be moved or copied from a DOS-environment because the user has no way of editing the "pointers" in the DEVTYPE.KEY file (see *Viewing Project Files from DOS*). Moving or copying a project file from a DOS-environment will scramble the correlations between the File Manager attribute configurations and the project files defined in the DEVTYPE.KEY file. If this happens, the File Manager attribute configurations will no longer correspond to the proper project files. If this occurs you must delete your entire database and rebuild it from scratch!

How to Copy a Project File to a Different Database:

1. Make certain that the root directory of the target database has been created from a DOS-window.
2. Load the desired project file.
3. Open the File Manager Setup dialogue box as described in *Setting the File Manager Options* earlier in this chapter. From the Database Directory window, designate the root directory of the target database. Click the dialogue box OK button.
4. ICS will now operate from the new database. Notice that the project file is still opened. Any changes made to the data or test setups cannot be written to the original project file.
5. From the toolbar or menu bar, select FILE/SAVE AS. This will open the File Manager. Specify the desired attributes in the corresponding attribute fields. While it is recommended that you specify the same attributes as the original project file, you can specify any attribute configuration you desire. In this case, specifying the same attribute configuration as the original project file will designate a unique project file, since ICS is operating from a different database.

6. If FILE/SAVE is selected instead of FILE/SAVE AS, the information will be written to the default project file. The default project file is identified by the designation "NONAME" in the ATTRIBUTE #1 field. No other attributes are defined.

Deleting Project Files

* ICS allows the user to delete either a single project file or all of the project files that have the same designation in the first attribute field.

A single project file is deleted by specifying all of the required project file attributes. All of the project files that share the same designation in the first attribute field are deleted by specifying the ATTRIBUTE #1 designation.

Deleting a Single Project File

A single project file is deleted using the delete function in the measurement mode menu bar. When a project file is deleted, all of the test setups, data, and hardware configurations contained in the project file will be lost.

* To delete only a test setup, please refer to Chapter 3: The Setup Editor/Deleting a Test Setup.

How to Delete a Single Project File:

1. Select FILE/DELETE DEVICE from the measurement mode menu bar. This will open the File Manager dialogue box.
2. Select the relevant alpha-numeric attribute designations that correspond to the project file in question. Click the attribute field scroll arrow to display a list of available options. Click the desired selection. After all the necessary fields have been identified, click OK.
3. The Delete Device dialogue box will appear in the middle of the screen displaying the attribute designations selected in Step #2. If these designations correspond to the correct project file, click OK; otherwise click CANCEL. Clicking OK will delete the designated project file.

Deleting a Family of Project Files

A project file is identified by specifying the appropriate combination of attribute designations in the File Manager dialogue box. The File Manager attribute fields are hierarchical in nature. This means that the first attribute field must be designated before the second, and the second before the third, the third before the fourth, and so on.
The File Manager was designed with the intent of using the attribute hierarchy to designate increasingly specific details about the project file. With this in mind, the first attribute field is

intended to identify a family of related project files. The designation in the first attribute field should relate to some characteristic that is common to each project file in the family, such as device type. Each additional attribute level is used to designate a more specific group of related project files until at last a single project file is identified by specifying the final attribute.

All of the project files associated with ATTRIBUTE #1 can be deleted in a single step by specifying the ATTRIBUTE #1 designation. If the File Manager is used as described, deleting all of the project files associated with ATTRIBUTE #1 will delete a family of related project files. However, the user can designate project files in any way desired, so the project files represented by a single ATTRIBUTE #1 designation may or may not be related.

How to Delete a Family of Project Files

1. Select FILE/DELETE A PROJECT from any menu bar. This will open the Delete Project dialogue box.
2. The Delete Project dialogue box will display a list of the ATTRIBUTE #1 designations associated with the current database. Select the desired ATTRIBUTE #1 from the list or manually designate the ATTRIBUTE #1 in the Project field. Paths cannot be specified in the Project field. You can only delete project files from the current database. If necessary, close the Delete Project dialogue box and change database directories as described in *Specifying the Database Root Directory* earlier in this chapter.
3. Click DELETE. All of the project files corresponding to the ATTRIBUTE #1 designation will be deleted.

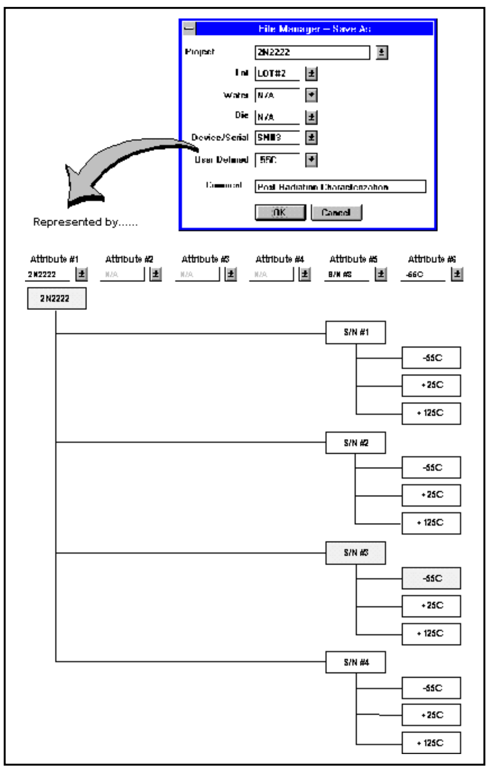


Figure 3-11: The ATTRIBUTE #1 Designation is Intended to Represent a Family of Related Project Files.

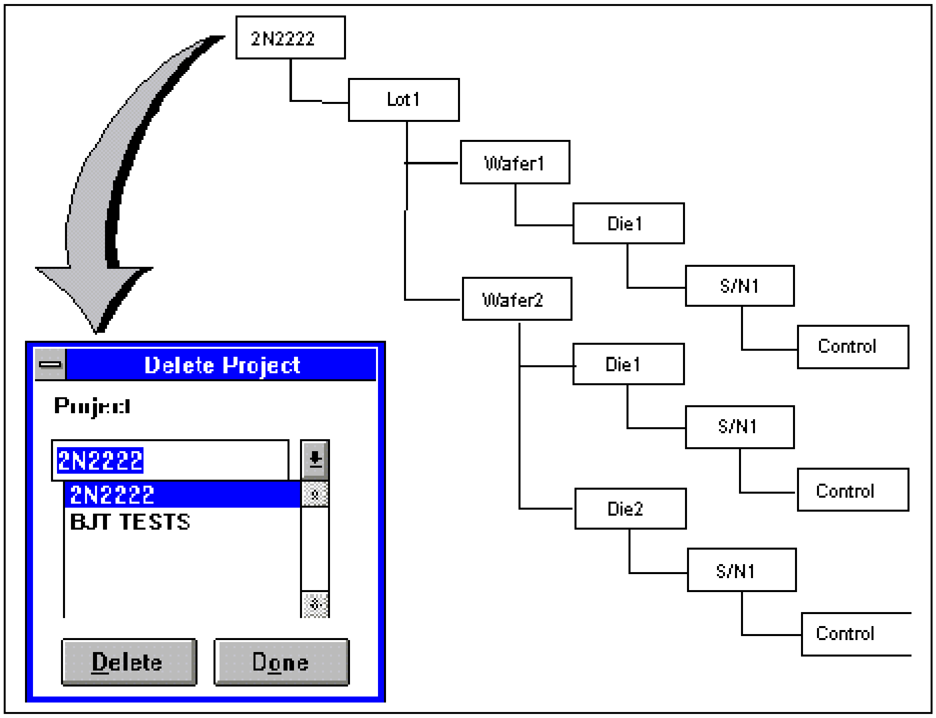


Figure 3-12: Selecting the 2N2222 ATTRIBUTE #1 Designation will Delete all of the Project Files Corresponding to the 2N2222 Attribute. In the Example Shown Here, Three Project Files will be Deleted.

Understanding the Temporary File

The test setups and data displayed in the desktop are not written to a project file nor is any corresponding project file updated until the user selects FILE/SAVE AS or FILE/SAVE. However, this information is recorded in a single temporary file prior to creating or updating a project file. The temporary file is updated each time the user creates or edits a test setup or measures data. When the user selects FILE/SAVE AS or FILE/SAVE, the information stored in the temporary file is copied to the corresponding project file.

* ICS operates with a single temporary file. The temporary file is named RESULTS.DAT and is located at the root of the designated database directory. RESULTS.DAT will always appear in the database root directory as long as ICS is running. When ICS is closed properly, the RESULTS.DAT file is destroyed.

The temporary file preserves the configuration of the desktop in the event of a sudden termination of the program. Any time a test setup is created or edited, the updated test setup and the present configuration of the remaining test setups on the desktop are written to the RESULTS.DAT file. Measured data is also preserved by copying data from memory to the RESULTS.DAT file every time data is sent from the instrument to ICS.

The temporary file is only destroyed when ICS is closed properly. The temporary file is not destroyed if ICS is terminated before the information in the temporary file is written to permanent storage by creating or updating a project file. Each time ICS is started, ICS searches for the RESULTS.DAT file in the database root directory. If the file is found, ICS prompts the user with a message box as soon as FILE/OPEN is selected. This message box asks whether or not the temporary file should be loaded. If you reply with a YES, the temporary file will be loaded preventing the loss of data. Once loaded, save the contents of the temporary file permanently by selecting the FILE/SAVE or FILE/SAVE AS function. If you answer NO, all of the information in the temporary file will be lost.

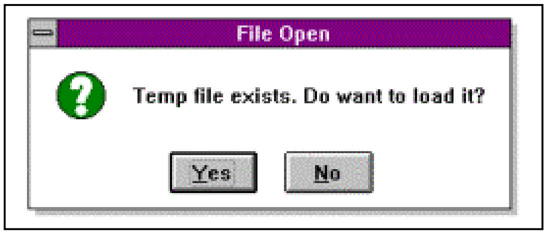


Figure 3-13: The Temporary File Prevents Data Loss.

Turning Off the Temporary File

* Implementing a temporary file in the way described above increases the time required to complete a device measurement.

Test time is increased because ICS will not release control to the user until after the data is copied from memory to the RESULTS.DAT file. Because the process of automatically copying data to a temporary file presents both an advantage and a disadvantage, the user is given the option of turning OFF the data component of the temporary file. Turning OFF the data component of the temporary file limits the content of the temporary file to the latest test setup configurations. The temporary file will no longer include any measured data. Data will no longer be copied from memory to disk each time data is sent from the instrument to ICS. Any measured data not written to a project file by periodically selecting FILE/SAVE AS or FILE/SAVE will be lost in the event of a sudden termination of the program. Despite the loss of measured data, the user will be able to measure new data if the temporary file is loaded, because the temporary file will still preserve the latest configuration of the test setups.

* As a default, ICS will copy measured data to the temporary file. This feature can be disabled by unselecting the corresponding switch in the Global controls dialogue box.

How to Stop Copying Measured Data to the Temporary File:

1. Open the Global Controls dialogue box by selecting the corresponding toolbar button. A project file must be opened before the Global Controls dialogue box can be displayed.
2. Turn OFF the Save Data to Temp switch at the top of the dialogue box.
3. Click OK to close the Global Controls dialogue box and restore control to the previously active window.

In some cases it may be advantageous to disable the data component of the temporary file. This is true during certain test setup configurations that implement the time domain utility in Time Measure mode. Disabling the data component of the temporary file will also improve the speed of the measurement process in Auto Export mode.
A test setup configured in Time Measure mode will measure device characteristics at designated time intervals while the device is stressed at a constant bias. The length of time required for the measured data to be copied from memory to the temporary file may be on the order of a few seconds depending upon the number of points measured. The minimum time interval that can be specified in Time Measure mode is 50ms. It is possible that the time required to copy the data from memory to the temporary file may prevent the specified time interval from being achieved. To prevent this situation, disable the data component of the temporary file when configuring a time measure test setup with a time interval of less than a few seconds.

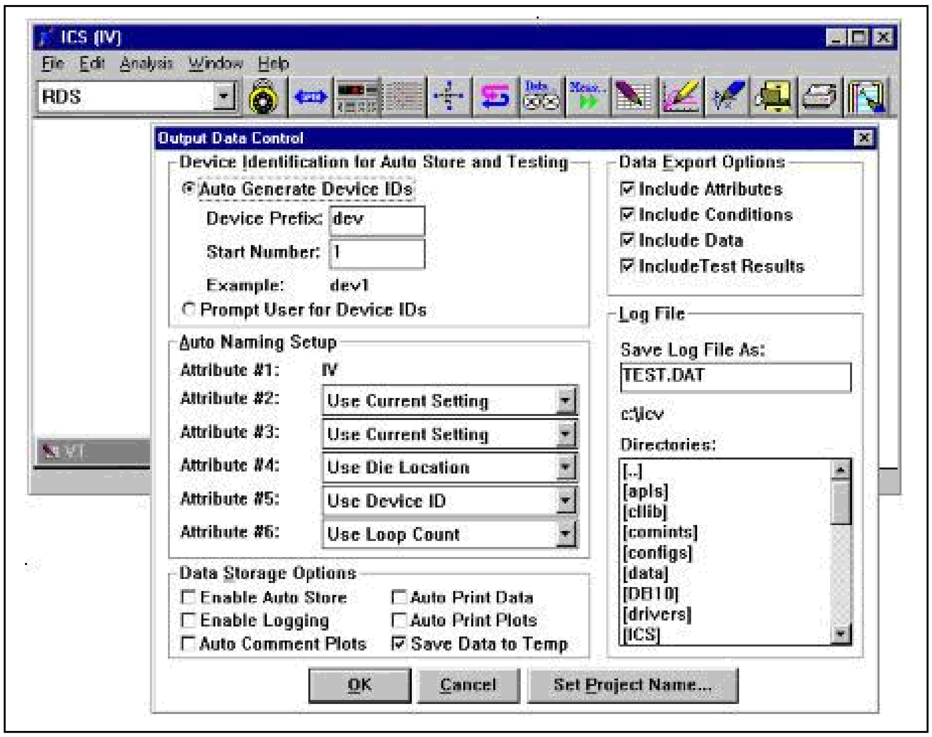


Figure 3-14: How to Stop ICS from Copying Measured Data to the Temporary File.

The Auto Export feature provides the capability to return data to an ASCII file. Auto Export is designed for users who want to generate a readable file of raw data. This is important if the user wishes to use the data outside of ICS. In applications such as these, the user may have no intention of saving any measured data to a project file. It is not necessary to include measured data in a temporary file if the data will not eventually be written to a ICS project file. If this is the case, disable the data component of the temporary file while in Auto Export mode in order to improve the speed of the measurement process.

* Disabling the data component of the temporary file does not mean that measured data cannot be saved to a project file. Measured data can always be saved to a project file by selecting FILE/SAVE AS or FILE SAVE.

This is true because data is always returned to the corresponding data window spreadsheets regardless of the temporary file configuration. Disabling the data component of the temporary file merely eliminates the ability to restore data to the desktop in the event that ICS is terminated unexpectedly. Any data that was previously written to an ASCII file will be unharmed.

Chapter 4: Measuring and Testing Devices

Performing Measurements

Measurement Types

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

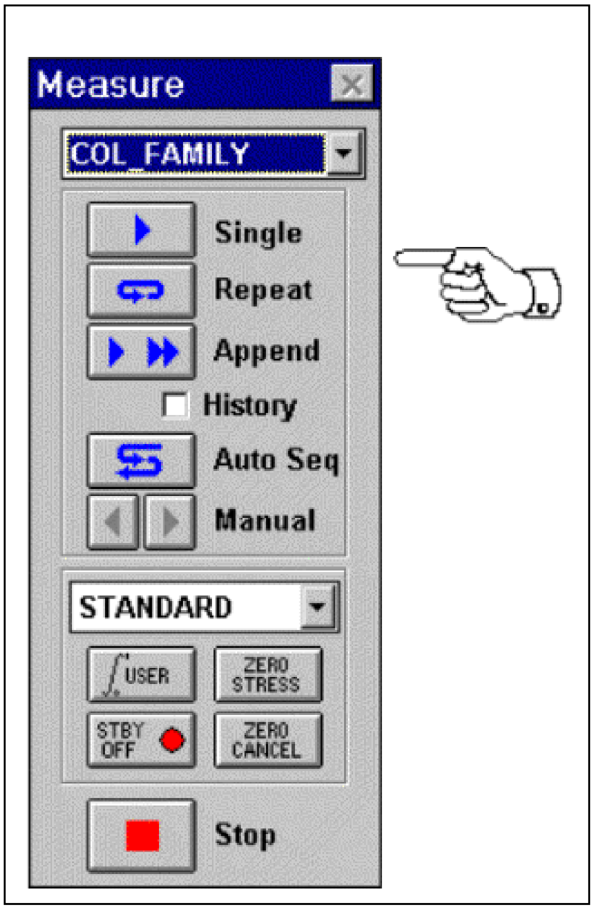


Figure 4-1: Measurement Types Available in ICS.

Single Measurement Type

Single measurements execute 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 measurements execute 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 measurements execute 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 measurements execute 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.

Manual Measurement Type

Manual measurements are only valid when the measurement mode is Interactive. The left manual measurement button executes a single point measurement for the current setup and attempts to bias the device at the previous point's bias values. The right manual measurement button executes a single point measurement for the current setup and attempts to bias the device at the next point's bias values. After completion of the single point measurement the data and plot windows are updated with the point's measured values.

Measurement Modes

ICS can be configured to execute standard, bias delay, time measure, real time, interactive, sequence bias, or sequence stress modes 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, Auto Seq, and Manual measurement types. Sequence Bias and Sequence Stress measurements are only executed during Auto Sequence measurement types.

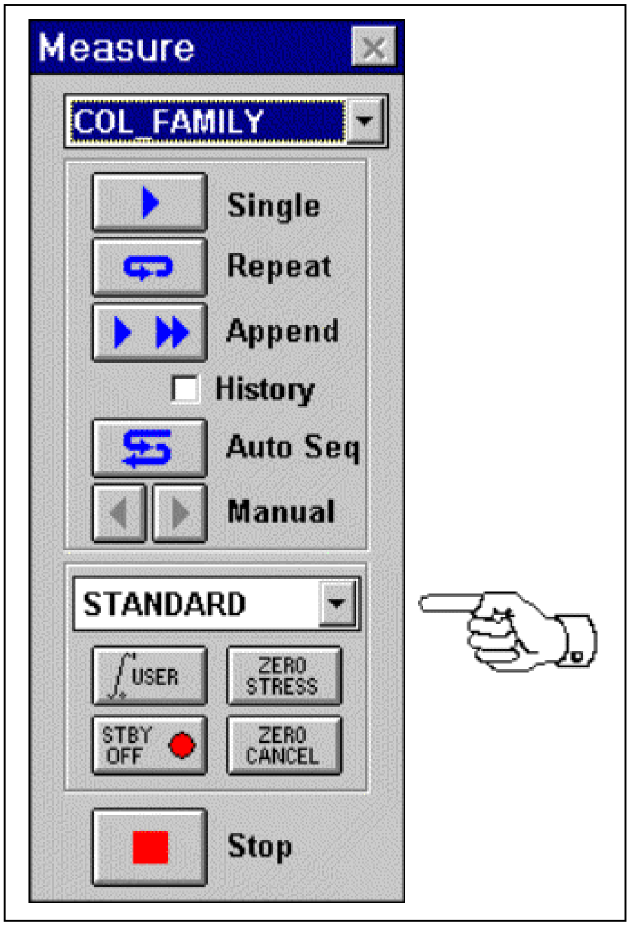


Figure 4-2: Measurement Modes Available in ICS.

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.*

Real Time Measurement Mode

Executes the standard measurement portion of the setup, one point at a time, as fast as possible and updates the data and plot windows as point measurements are executed.

Interactive Measurement Mode

Executes the standard measurement portion of the setup, one point at a time. The execution of a point measurement is initiated by the selection of either the left or right manual measurement type buttons. The left manual measurement button executes a single point measurement for the current setup and attempts to bias the device at the previous point's bias values. The right manual measurement button executes a single point measurement for the current setup and attempts to bias the device at the next point's bias values. After completion of the single point measurement the data and plot windows are updated with the point's measured values.

Sequence Bias Measurement Mode

Executes the standard portion of the setup as a power supply to be used with the other setups selected to be executed in a sequence. The Sequence Bias setup will cause the associated instrument to remain biased during the execution of all other setups in the sequence. *Refer to Creating a Time Domain Test Setup in chapter 2 for a description of Sequence Bias Measurements.*

Sequence Stress Measurement Mode

Executes the standard portion of the setup in a time based stress measurement manner. This measurement mode allows for the stressing of a device followed by multiple IV and/or CV measurements defined as a sequence measurement. The stress-measure sequence may be repeated by defining multiple stress periods for the Sequence Stress setup. *Refer to Creating a Time Domain Test Setup in chapter 2 for a description of Sequence Stress Measurements.*

Interactive Measurement Controls

ICS can interactively change Measurement Integration, Measurement Standby mode, perform Zero Cancel functions, and Zero the accumulated stress from the Measurement Remote control dialog by a press of the corresponding button. These functions must be supported by your instrumentation.

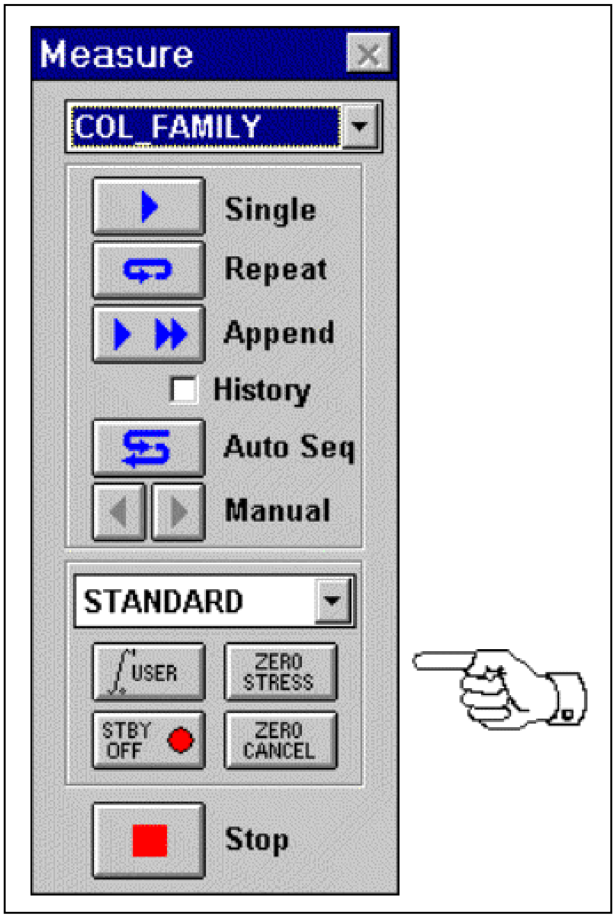


Figure 4-3: Interactive Measurement Controls Available in ICS.

Executing a Test Setup

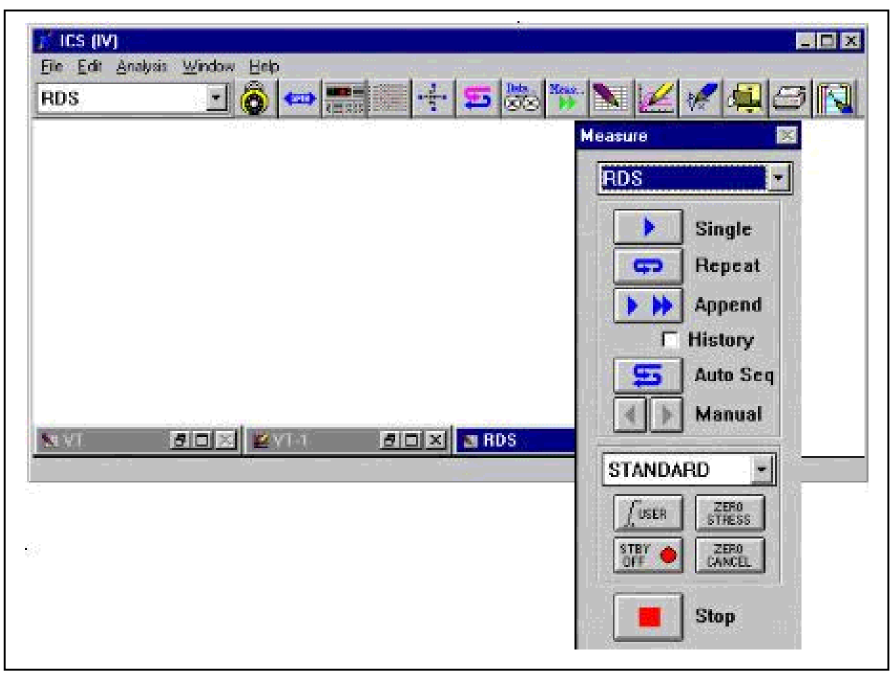


Figure 4-4: How to Execute a Test Setup.

How to Execute Test Setup:

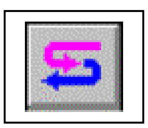
1. Click the toolbar Measure button to display the Measurement Remote Control.
2. Select the active test setup from the Setup list combo box located at the top of the Measurement Remote Control.
3. Select the desired measurement mode from the measurement mode combo box.
4. Click on the measurement type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

Executing a Sequence of Test Setups

ICS 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, ICS will execute the test setup sequence identified in the Measure Sequence dialogue box.

Defining a Test Setup Sequence

* ICS can store and execute any combination of test setups as long as all of the test setups are selected from the same project file. ICS can store one test sequence definition per project file.



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.

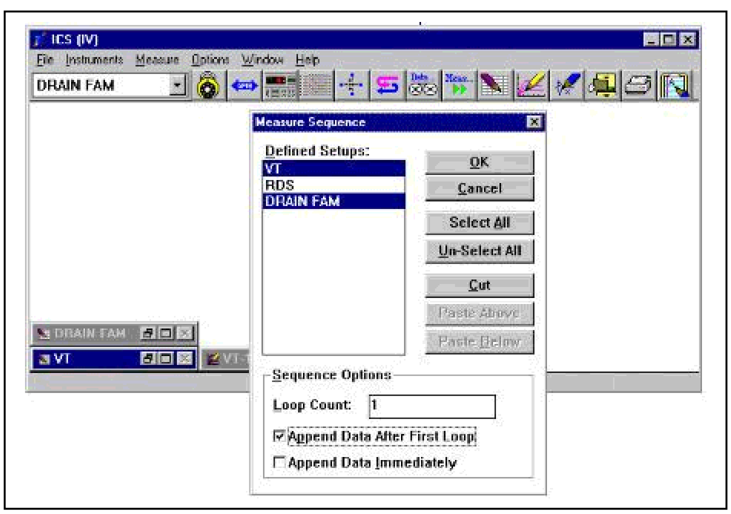


Figure 4-5: How to Define a Test Setup Sequence.

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. ICS 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 ICS 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

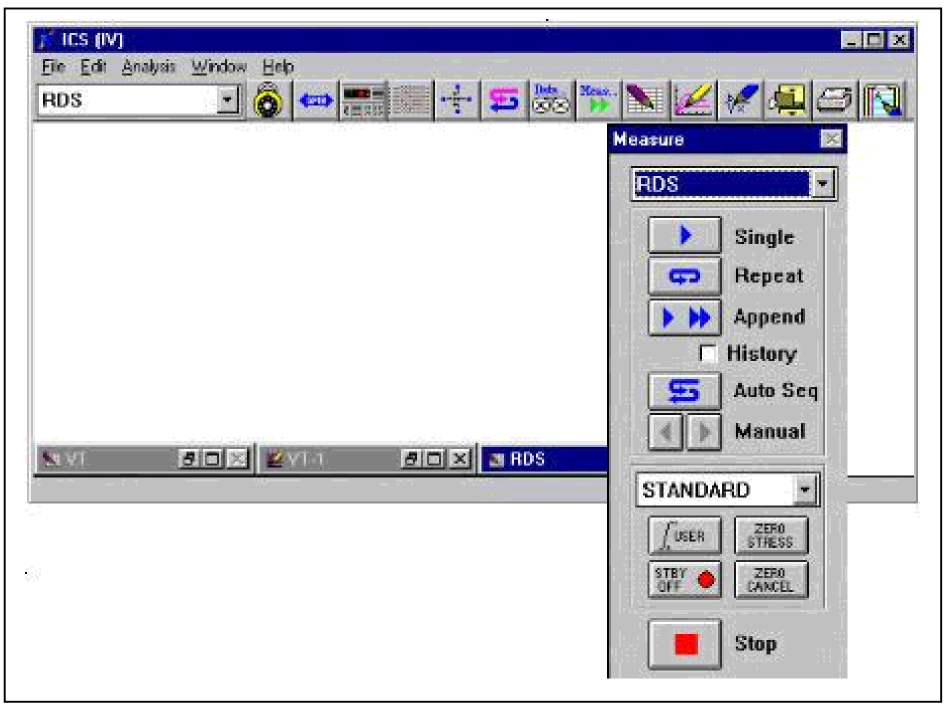


Figure 4-6: How to Execute Sequential Test Setups.

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.

* When ICS is configured in Sequence Measure mode, the active test setup corresponding to Single Measure mode will remain displayed in the toolbar Setup Window. This display is ignored when the test sequence is executed. The sequence will be executed as defined in the Sequence Measure dialogue box.

Using the Auto-Store Function

* The Auto-Store function provides the capability to test an entire device lot, or any group of identical devices, while storing the data for each device in a unique project file.

When the Auto-Store function is active, measurement results will be written to a unique project file every time the test instrument is actuated. The Auto-Store function creates a new project file by automatically incrementing the numerical value of the Device/Serial#, or by using the probe die location, switch scan level, or the sequence loop count while copying the contents of the remaining attribute fields from a specified master file to the new project file.

Since the project files generated with the Auto-Store function are based upon a master project file, the need to configure a test setup sequence allows the user to generate a family of project files that consist of a selected group of test setups rather than the entire desktop. (It is still possible to include every project file test setup in the generated project files by implementing a test sequence that is comprised of the entire desktop.)

The Auto-Store function creates a new project file each time a measurement is executed. Each project file created with Auto-Store will have all of the project file attribute fields in common, except for the designated unique fields.

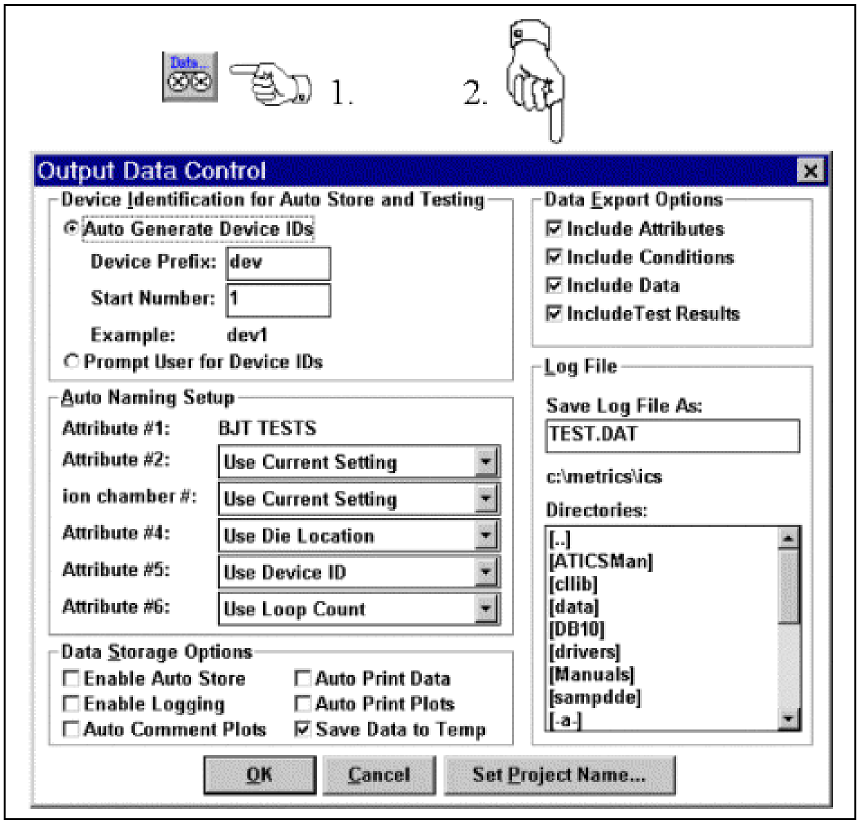


Figure 4-7: The Global Control Dialogue Box.

The Auto-Store controls are located in the Global Control dialogue box. The Global Control dialogue box is opened by selecting the GLOBAL CONTROL toolbar button.

Device Identification for Auto Store and Testing

The Device Identification controls allow the user to either be prompted for a device identification or allows ICS to automatically generate a device identification based upon a prefix and base number. Auto generation of device identification occurs by incrementing the base number and appending the base number to the prefix for each measurement.

Auto Naming Setup

The Auto Naming controls allow the user to specify what data information is to be used for each attribute when the auto store feature is being used. Attribute 1 always uses it's current file manager setting. All other attributes may be set to use one of the following:

Use Current Setting- Current file manager setting.

Use Device ID- Manual or auto generated device ID.

Use Device Location- Prober die location.

Use Test Result- Global Pass/Fail test result

Use Loop Count- Sequence loop count.

Use Switch Scan Level- Switch scan number.

Data Storage Options

The Data Storage Options allow the user to customize how and when data is stored or printed.

Enable Auto Store- Turns the auto store feature on or off.

Enable Logging- Turns the ASCII log feature on or off. See Chapter 7, ASCII Auto Export for a full description of this feature.

Auto Comment Plots- Automatically comment plots after measurement execution using the current attributes.

Auto Print Data- Automatically print data views after measurement execution

Auto Print Plots- Automatically print plots after measurement execution

Save Data to Temp- Saves data to a temp file prior to executing measurements.

Data Export Options

The Data Export Options allow the user to customize the data output when exporting ASCII data and auto logging ASCII data.

Log File

The Log File controls describe the location of the ASCII log file.

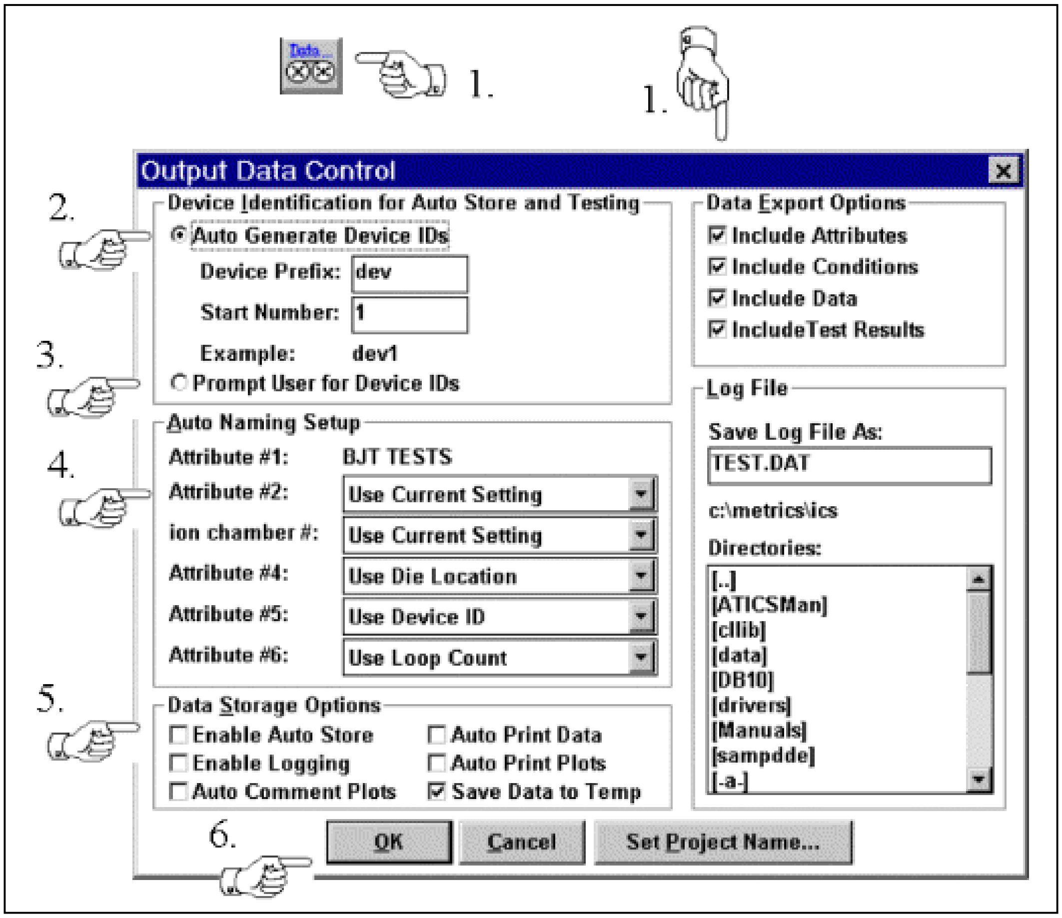


Figure 4-8: How to Use the Auto-Store Function.

How to Use the Auto-Store Function:

1. Click on the GLOBAL CONTROL (DATA) button on the toolbar to display the Global Control dialogue box.
2. If automatic ID generation is desired, enter an alphanumeric prefix in the DEVICE PREFIX window. The designation in the DEVICE PREFIX window will preface the incremental value written to the Device/Serial# field in every project file generated by the Auto-Store function. The designation entered in the DEVICE PREFIX window will immediately appear in the read-only attribute field of the Global Control dialogue box. Enter the serial number of the first device in the STARTING NUMBER window. This is the value at which the Auto-Store function will begin incrementing the Attribute #6 field each time a test setup or test sequence is executed.
3. If user prompting of IDs is required select the Prompt User option.
4. Specify the data parameters to be used for each of the file manager attributes.
5. If the contents of the attribute fields are unacceptable for the attributes designated to Use Current Setting, click the CHANGE PROJECT NAME button. This will open the File Manager dialogue box. Specify new settings for all attributes designated as Use Current Setting. Click OK to close the File Manager dialogue box and update the read-

only designations in the Global Control dialogue box with the specifications just entered.

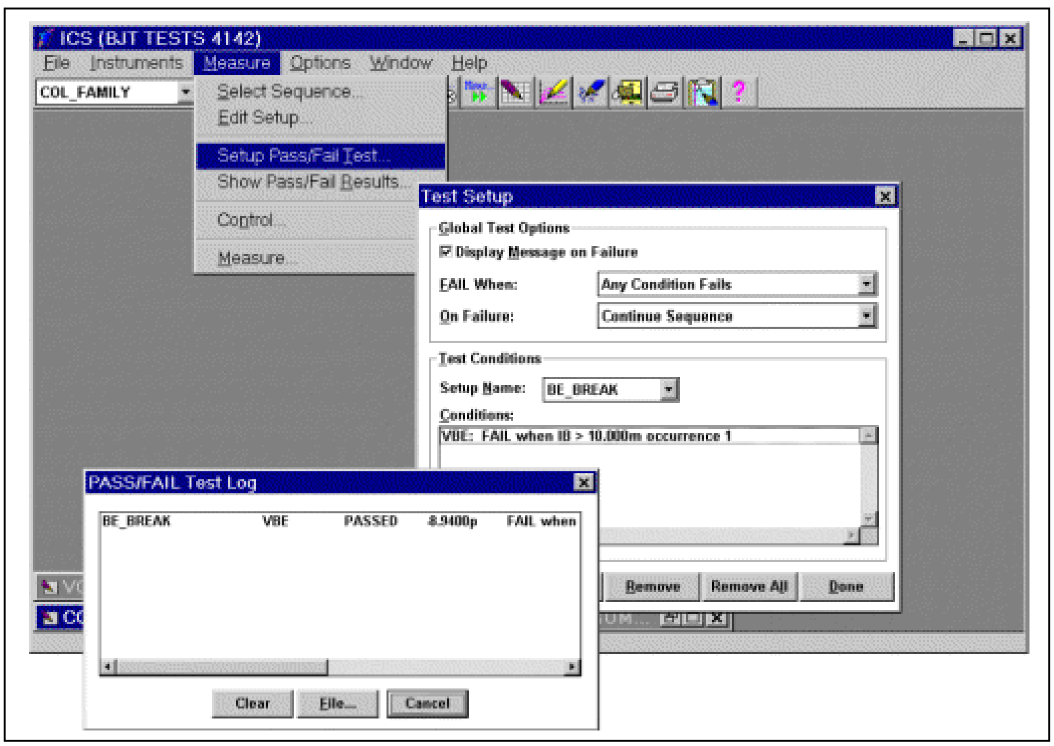
6. Activate the Auto-Store function by clicking on the Auto-Store switch at the top of the dialogue box.
7. Click OK to close the Global Control dialogue box and restore control to the ICS desktop.

Each time you execute a test setup or test sequence while the Auto-Store function is active, ICS will store the data and corresponding test setup(s) to a separate project file. Each project file will be uniquely identified by an incremental value in the project file's Attribute #6 field. The remaining attribute fields will include the same attribute designations specified in the Global Control dialogue box.

To terminate the Auto-Store function, open the Global Control dialogue box and turn off the Auto-Store switch.

Performing Pass/Fail Testing

Interactive Characterization Software(ICS) can be configured to execute Pass/Fail tests which are based upon the comparison of data values against test thresholds. Multiple Pass/Fail test may be configured for a single setup. ICS also allows the flow of a sequence measurement to be interrupted based upon Pass/Fail test results.



Configuring Pass/Fail Tests

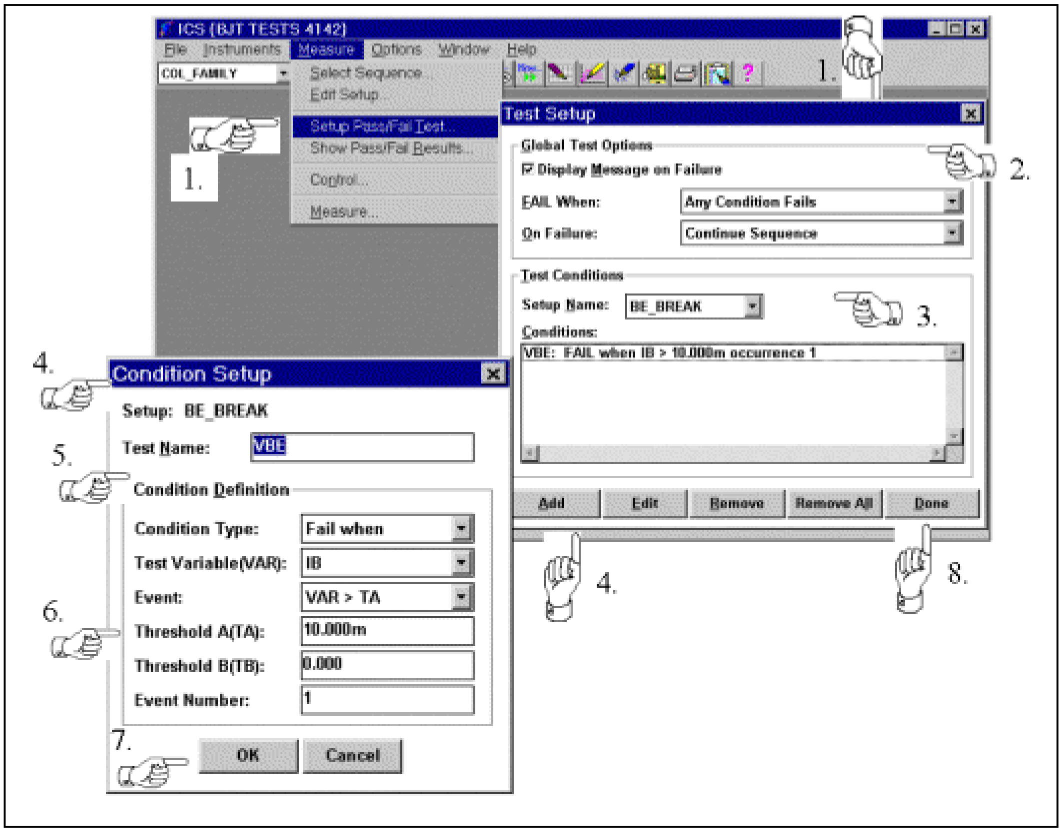


Figure 4-9: How to Use Configure a Pass/Fail Test.

How to Configure a Pass/Fail Test:

1. Click on the Measure/Setup Pass/Fail Test menu items to display the Test Setup dialogue box.
2. Specify the Global Test options. These options apply to all tests for all setups.
3. Select the target setup for the test from the Setup Name control.
4. Press the Add button to display the Condition Setup dialog box.
5. Enter a name for the test in the Test Name control.
6. Select the data variable on which the test is to be performed from the Data Variable control.
7. Select the desired test event to be used from the Event control.
8. Specify the A and B thresholds if required in the Threshold A and Threshold B controls.
9. Specify the event number on which the condition should be considered a failure in the Event Number control.
10. Press the OK button to complete the Condition Setup.
11. Press the OK button to complete the Test Setup.

Displaying Pass/Fail Tests Results

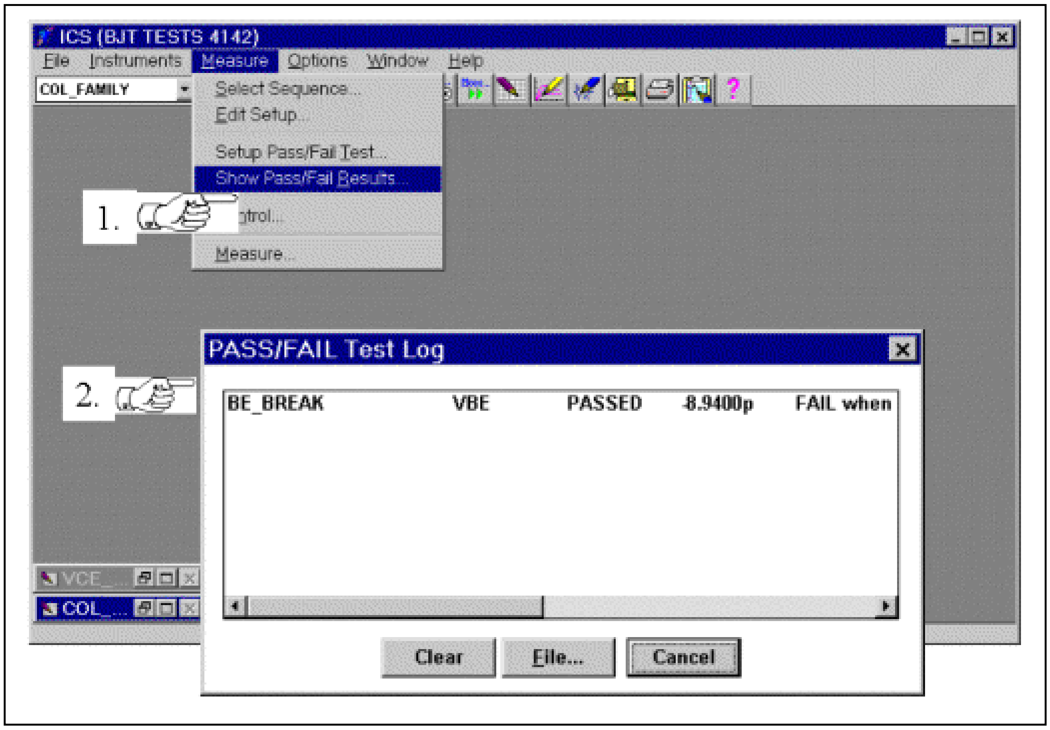


Figure 4-10: How to Display Pass/Fail Test Results.

How to Display Pass/Fail Test Results:

1. Click on the Measure/Show Pass/Fail Results menu items to display the Pass/Fail Test Log dialogue box. The dialog will remain displayed until the Cancel button is pressed. ICS may be used while the dialog is being displayed.
2. Current test results are displayed in the log window.
3. Press the Clear button to clear the contents of the log window.
4. Press the File button to write the test results to a file.
5. Press the Cancel button to close the log window.

Chapter 5: Data Windows and Data Analysis

What is a Data Window?

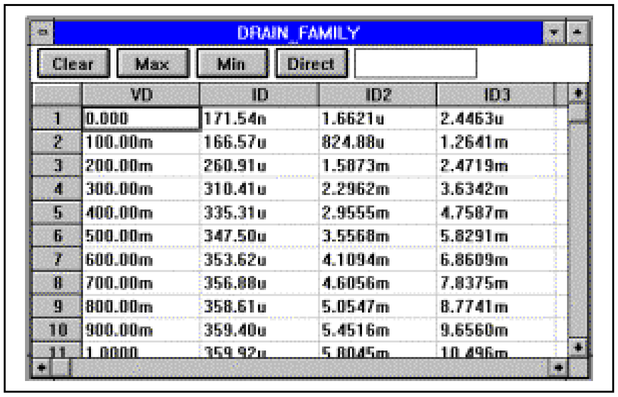


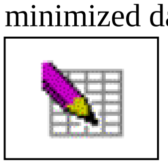
Figure 5-1: Example of a Data Window Spreadsheet.

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 ICS desktop. *Minimizing* a data window will reduce the window to an icon positioned along the base of the desktop. A minimized data window is represented by a white icon along the bottom 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.

Learning 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 WindowsSYMBOL 226 \f "Symbol" 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. To test a sample of identical devices and preserve the data for each device, refer to *Chapter 4: Measuring Devices/Using the Autostore Function.*

Editing Data Window Spreadsheets

Setting Data Precision

The precision of the numerical data stored in a ICS 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 ICS 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 ICS.INI file. Every data window will be created and displayed at the precision identified in the ICS.INI file independent of project file.

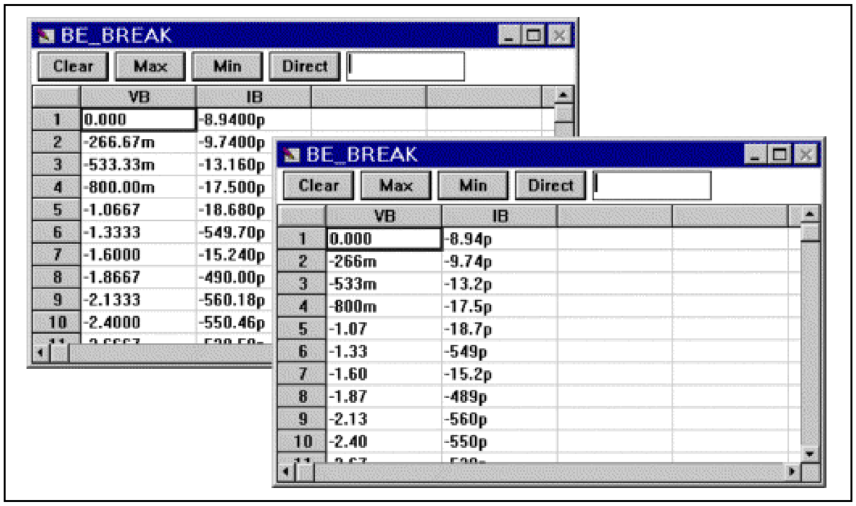


Figure 5-3: The Precision of the Numerical Data Displayed in a Data Window Spreadsheet can be Controlled by the User.

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 ICS. 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.

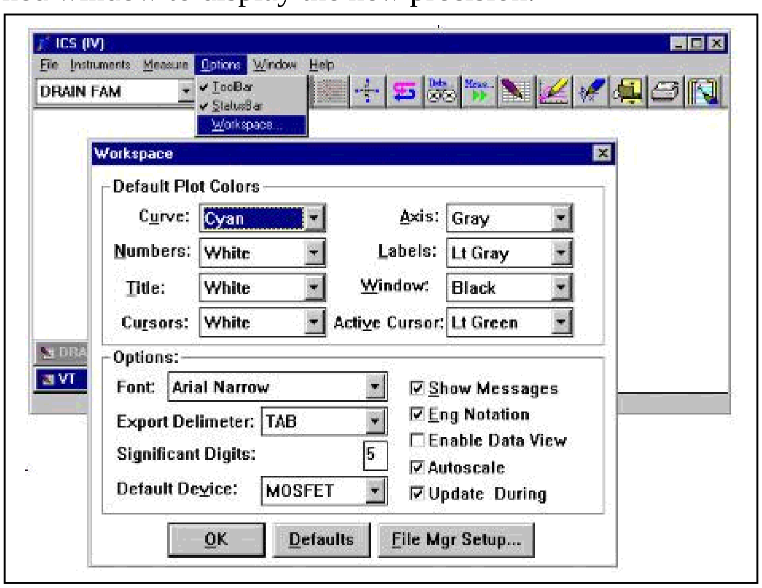


Figure 5-4: How to Designate Spreadsheet Precision

Designating Data Window Content

The list of data vectors which are displayed in the Data Window, as well as saved to the project file, may be changed by changing the Data View of the Data Window. The Define Data View dialogue box is used to edit the list of data vectors which are displayed in the Data Window and saved to the project file.

* The Data View provides the user with the capability to define data vectors to be displayed in the Data Window and saved to the project file. Note that ONLY the data view vectors will be saved in the project file.

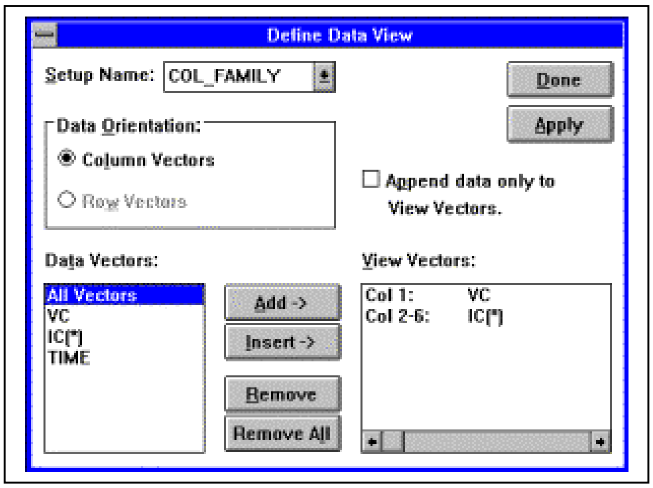


Figure 5-5: The Data View provides the User with the Capability to Designate Data Window and Project File Contents.

The Define Data View Dialogue Box

A Data View, for a given setup, is a list of data vectors which will be displayed in the setups Data Window and when the project is saved only the data vectors in the Data View will be saved to the project file. The primary feature of the Data View is the ability to only view and save the desired data vectors.

Setup Name

Identify the test setup which will be the target of the data view configuration from the SETUP NAME control.

Data Vectors

The Data Vectors list box contains a list of all available data vectors for the selected test setup.

Append Data to only to View Vectors

Enabling this option causes data to be appended to only the vectors listed in the View Vectors list during append measurements. If used this can significantly increase the number of append measurements which may be performed.

View Vectors

The View Vectors list box contains a list of all of the data vectors which will be displayed in the Data Window and saved to the project file.

Edit Buttons

The Edit Buttons, including ADD, INSERT, REMOVE and REMOVE ALL, are used to edit the contents of the VIEW VECTORS list box.

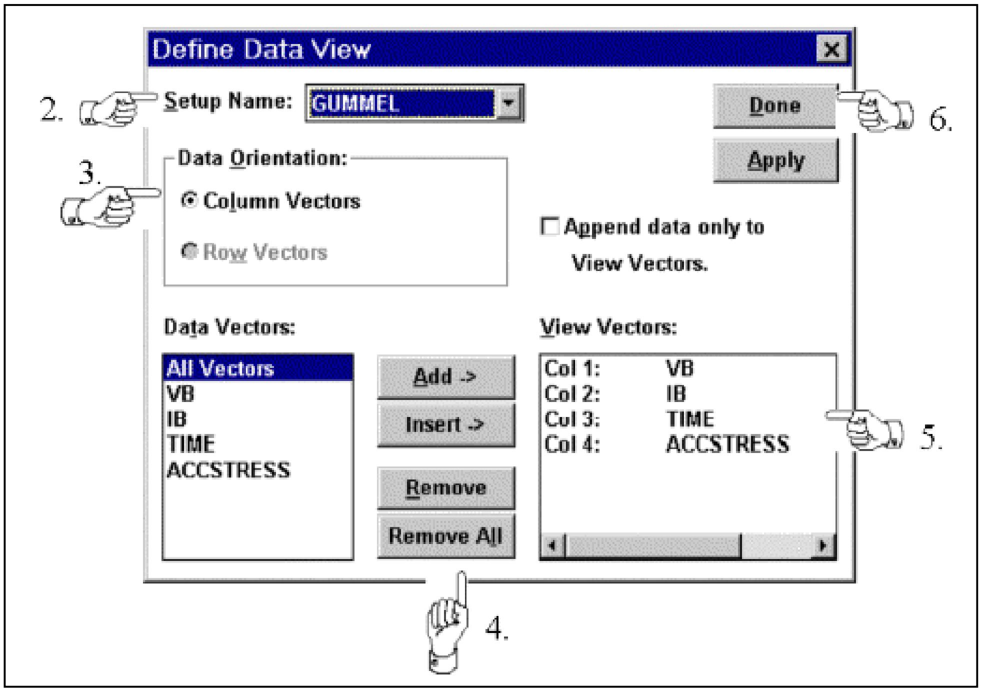


Figure 5-6: How to Add Data Vectors to a Data View.

How to Add Data Vectors to a Data View

1. Before defining a Data View the data view functionality must be enabled for ICS. To enable the data view function select the OPTIONS/WORKSPACE menu items to display the Workspace dialogue box. Turn on the ENABLE DATA VIEW switch in the OPTIONS control group and the Data View functionality will be enabled.
2. Select the data view source test setup from the SETUP NAME control.

3. Highlight the data vector name, to be added to the view vectors list, in the DATA VECTORS list. Highlight ALL VECTORS and all data vectors will be added to the view vectors list.
4. Click on the ADD button.
5. Click on DONE to complete the data view definition. Click on APPLY to view the data view definition results in the data window.

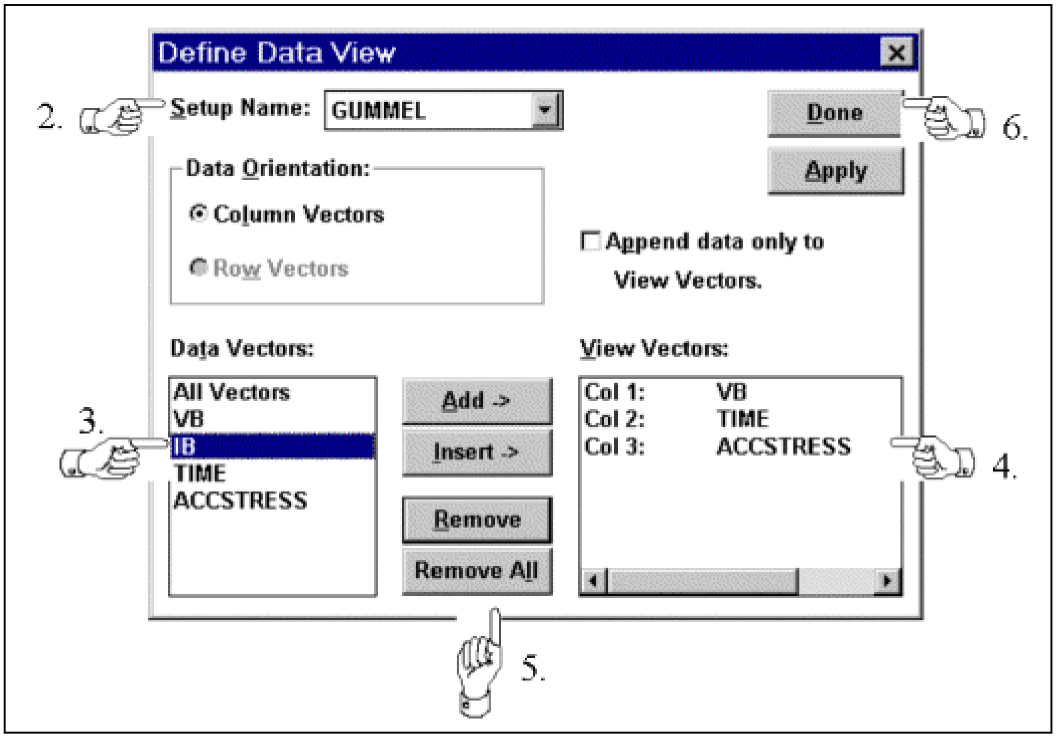


Figure 5-7: How to Insert Data Vectors into a Data View.

How to Insert Data Vectors into a Data View

1. Before defining a Data View the data view functionality must be enabled for ICS. To enable the data view function select the OPTIONS/WORKSPACE menu item to display the Workspace dialogue box. Turn on the ENABLE DATA VIEW switch in the OPTIONS control group and the Data View functionality will be enabled.
2. Select the data view source test setup from the SETUP NAME control.
3. Highlight the data vector name, to insert in the view vectors list, from the DATA VECTORS list.
4. Highlight the data vector name in the VIEW VECTORS list where the new data vector is to be inserted.
5. Click on the INSERT button.
6. Click on DONE to complete the data view definition. Click on APPLY to view the data view definition results in the data window.

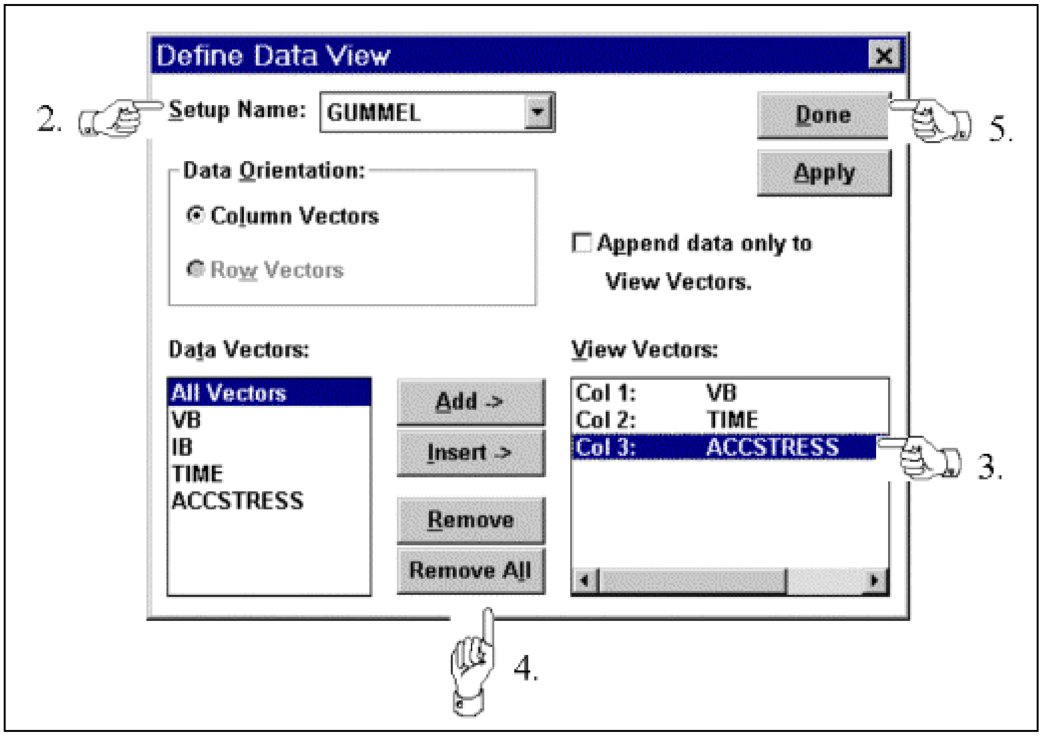


Figure 5-8: How to Remove Data Vectors from a Data View.

How to Remove Data Vectors from a Data View

1. Before defining a Data View the data view functionality must be enabled for ICS. To enable the data view function select the OPTIONS/WORKSPACE menu item to display the Workspace dialogue box. Turn on the ENABLE DATA VIEW switch in the OPTIONS control group and the Data View functionality will be enabled.
2. Select the data view source test setup from the SETUP NAME control.
3. Highlight the data vector name, to be removed from the view vectors list, in the VIEW VECTORS list.
4. Click on the REMOVE button. Click on the REMOVE ALL button to remove all data vectors from the VIEW VECTORS list.
5. Click on DONE to complete the data view definition. Click on APPLY to view the data view definition results in the data window.

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".

* 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.

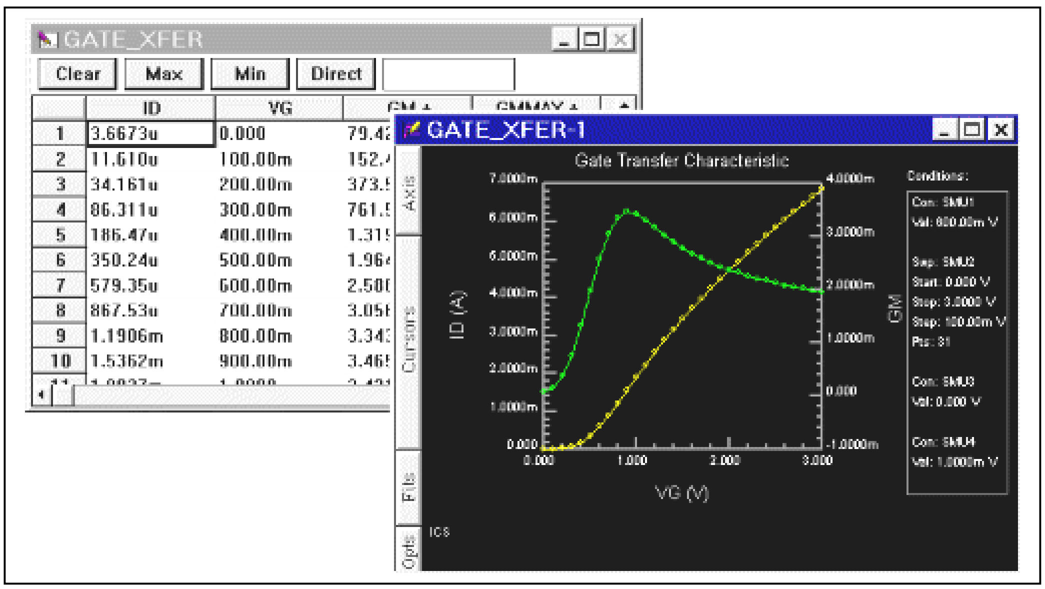


Figure 5-9: The Transform Editor provides the User with the Capability to Define Data Vectors that are Functions of Measured Data.

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:

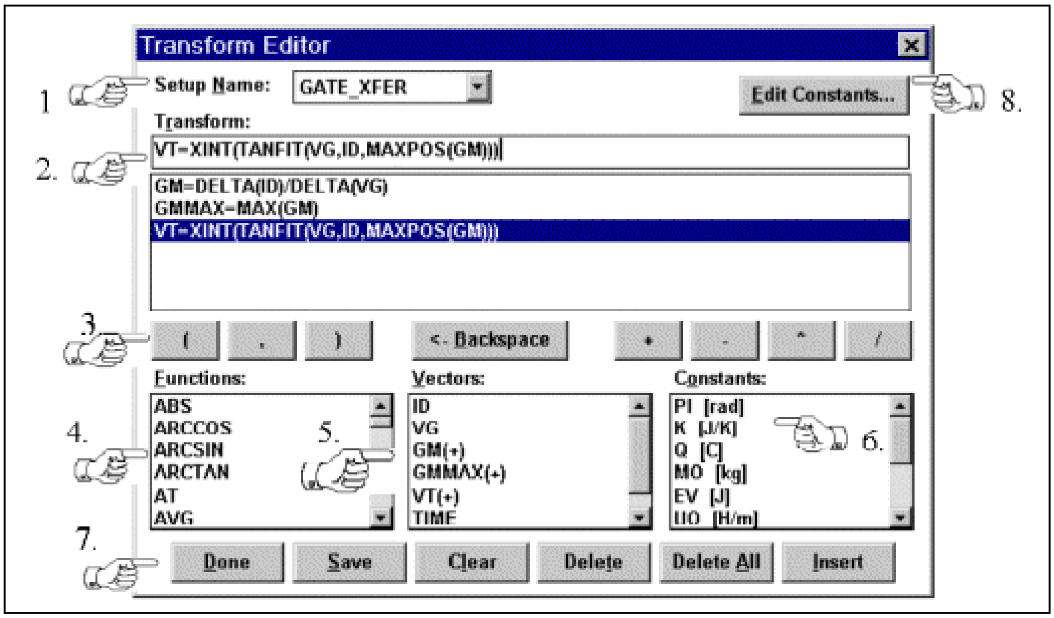


Figure 5-10: The Transform Editor dialogue box.

1. **Setup Name:** The Setup Name combo box is used to select the setup in which the transform will be defined.
2. **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.
3. **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.
4. **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.

5. **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.
6. **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.
7. **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.

8. **Edit Constants Button:** The Edit Constants button displays the Constant Editor dialogue box. The Constant Editor dialogue 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.

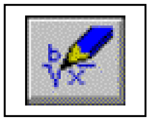
Transform Editor Operation

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, icluding 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. ICS 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.

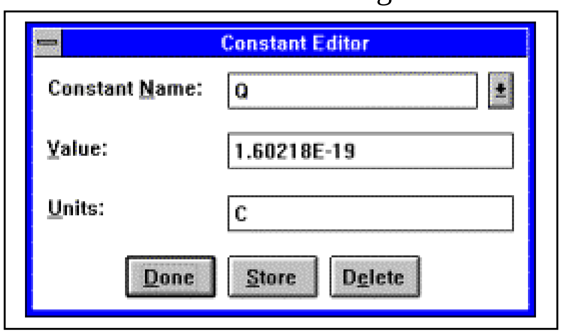


Figure 5-11: The Constant Editor Dialogue Box Provides the Capability to Edit, Add, or Delete Constants from the Constants List Box.

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 (Vth) extraction.

Transform Expression	Function
GM=DELTA(ID)/DELTA(VG)	Calculates GM
GMMAX = MAXPOS(GM)	Finds the position of the maximum value of GM
GMTFIT = 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 = XINT(GMTFIT)	VTH equals the x intercept of GMFIT

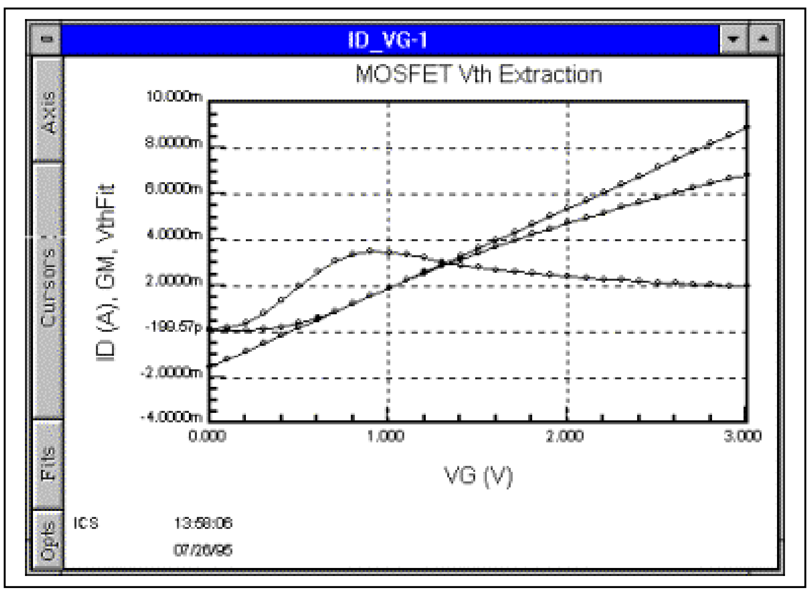


Figure 5-12: Plot of ID, GM, and GMFIT versus VG.

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

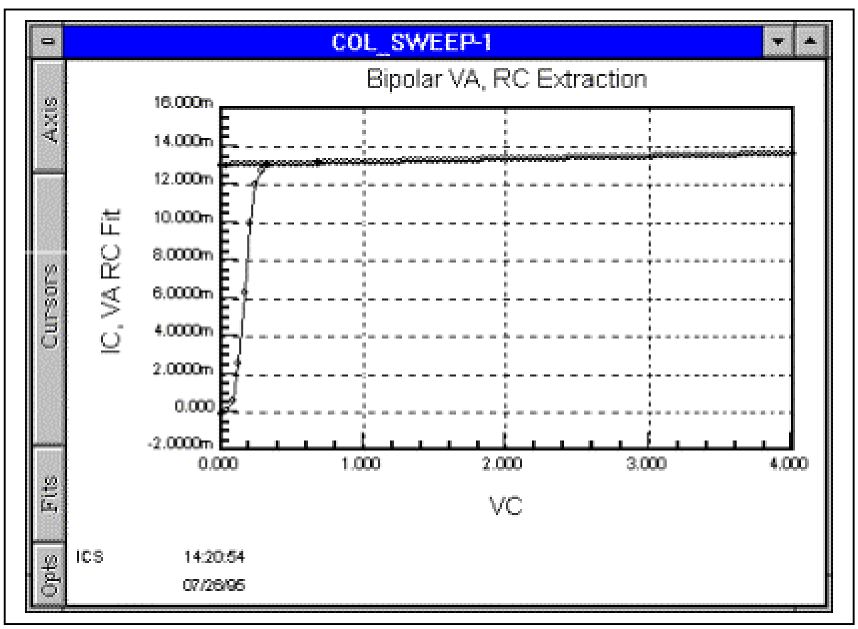


Figure 5-13: Plot of IC and VA RC Fit versus VC.

MOSFET Subthreshold 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

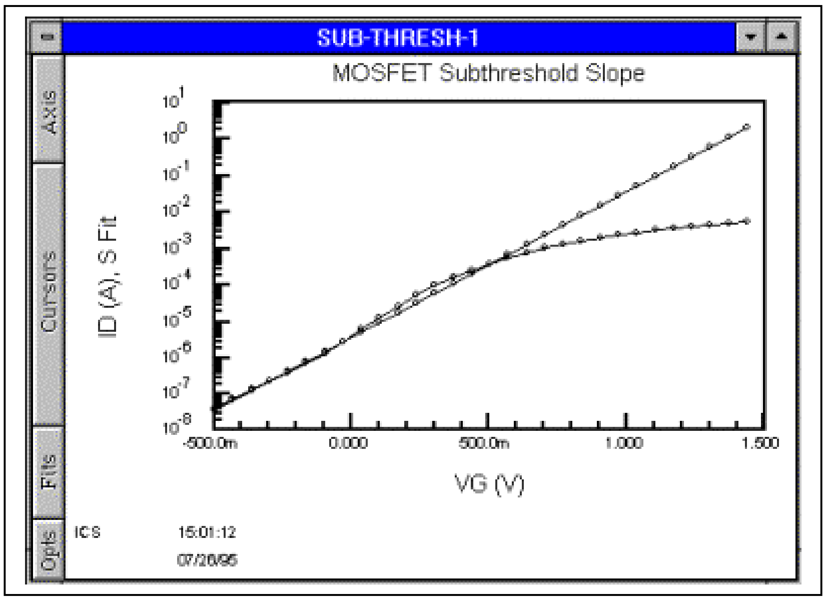


Figure 5-14: Plot of ID and S Fit versus VG.

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

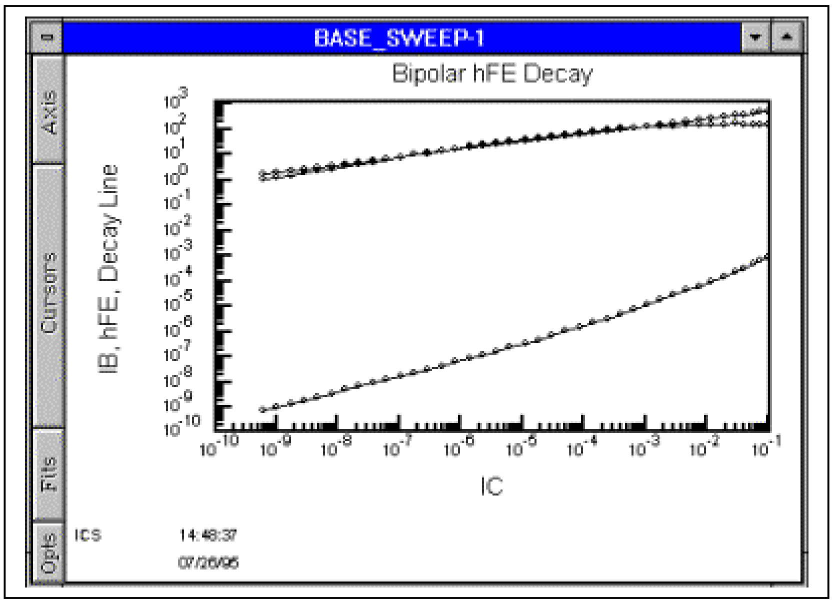


Figure 5-15: Plot of IB, hFE, and Decay Fit versus IC.

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=(G^2+w^2*C^2)C/(a^2+w^2*C^2)-OPENVAL where w=2*pi*FREQ a=G-(G^2+w^2C^2)*Rseries
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 >= B, return D as a single value.
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 >= B, return D.
SYNTAX:	[VECTOR | REAL | LONG] = CONDPP (A [VECTOR | REAL | LONG], B [VECTOR | REAL | LONG], C [VECTOR | REAL | LONG], D [VECTOR | REAL | LONG])
EXAMPLE:	TEST=CONDPP(IB,IC,IB,IC)

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 = coeffa * exp (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.

GCORR=(G^2+w^2*C^2)a/(a^2+w^2*C^2)

where

w=2*pi*FREQ

a=G-(G^2+w^2C^2)*Rseries

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 indicies 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=MAX(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 folowing input parameteres and equations. (See HP Application Note 322 (Analysis of Semiconductor Capacitance Characteristics). This method assumes that 1/Cox is negligible with respect to 1/Cmin.

Wmax_nm: Max depletion layer width(nm).

NI: Intrinsic carrier concentration/cm3.

T: Temperature (K).

Wmax-nm=Esi*Eo*A/Cmin

Fermi = k*T/q*ln(Nsub/ni)

Nsub = 4*Fermi*Eo*Esi/[q(Wmax_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 folowing input parameters and equations. (See MOS (Metal Oxide Semiconductor) Physics and Technology by Nicollan and Brews). This method does not assume that 1/Cox << 1/Cmin.

CSMIN: Min depletion layer C (F).

CSMAX: Max depletion layer C (F).

AREA: Gate area (cm2).

T: Temperature (K).

Wmax = Esi * Eo * area * (1/cmin - 1/cmax)

Fermi = (k*T/q)*ln(Nsub/ni)

Inversion = 2.0 * Fermi

Nsub = 2 * Eo * Esi * Inversion/(q * Wmax^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 = coeffb * log (x) + 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 = coeffb * log(x) + 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 = coeffb * log(x) + 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 = coeffb * log(x) + 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 = coeffa * exp [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 = coeffa * exp [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 = coeffa * exp [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 = coeffa * exp [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 = coeffa * exp(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 = coeffa * exp(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 = coeffa * exp(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 = coeffa * exp(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.

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

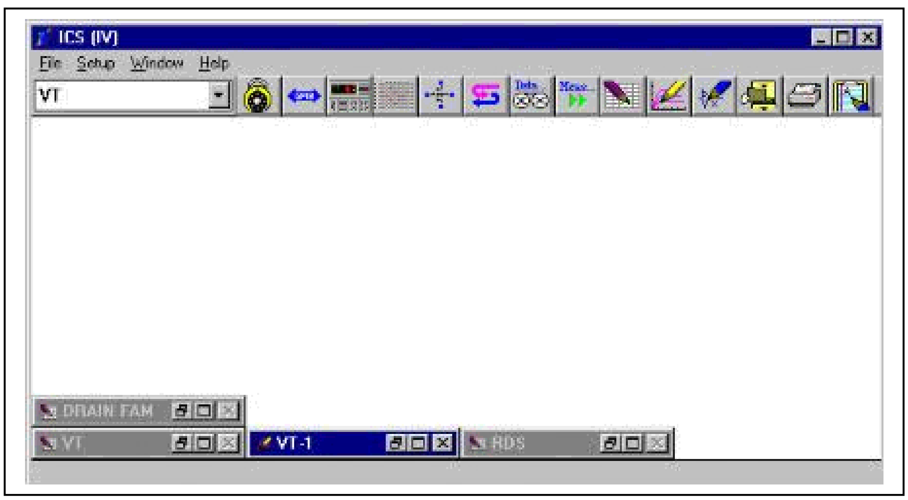
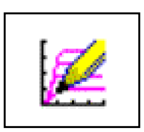


Figure 6-1: The blue (dark) icons shown in this view are minimized plot windows.



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.

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 times 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 ICS. 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 ICS 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 ICS 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 ICS:

Axis

Y_1/Y_2 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. Y_1-Axis Label
8. Y_2-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
	Y_1 -Axis Display	Linear
	Y_2 - 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

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, ICS 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.

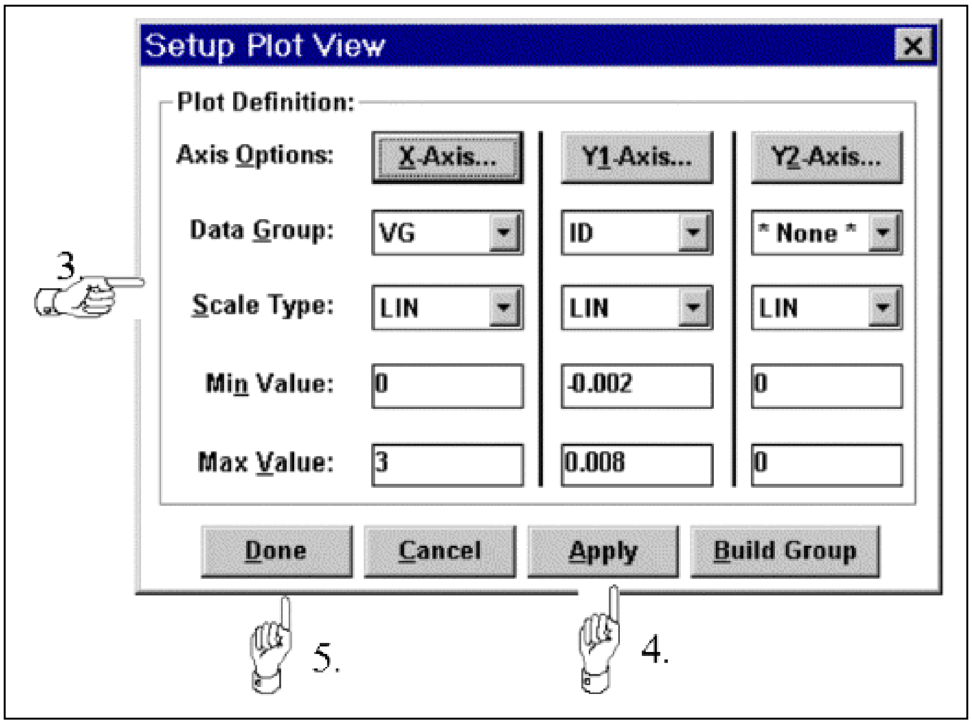


Figure 6-2: How to Create a New Plot

How to Create a New Plot:

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.

1. 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.
2. 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.

3. Click on the Apply button. This will create the plot window without closing the Plot Data dialogue box. When ICS 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.
4. 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 up-dated 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 all users to plot more than one data vector against the Y1 and Y2 axis.

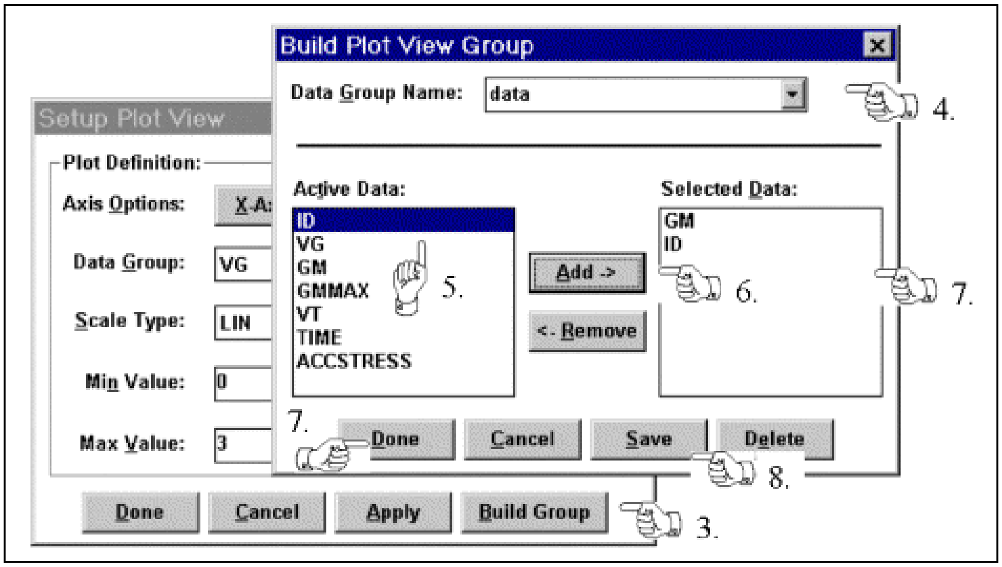


Figure 6-3: How to Create a Plot Data Group.

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.

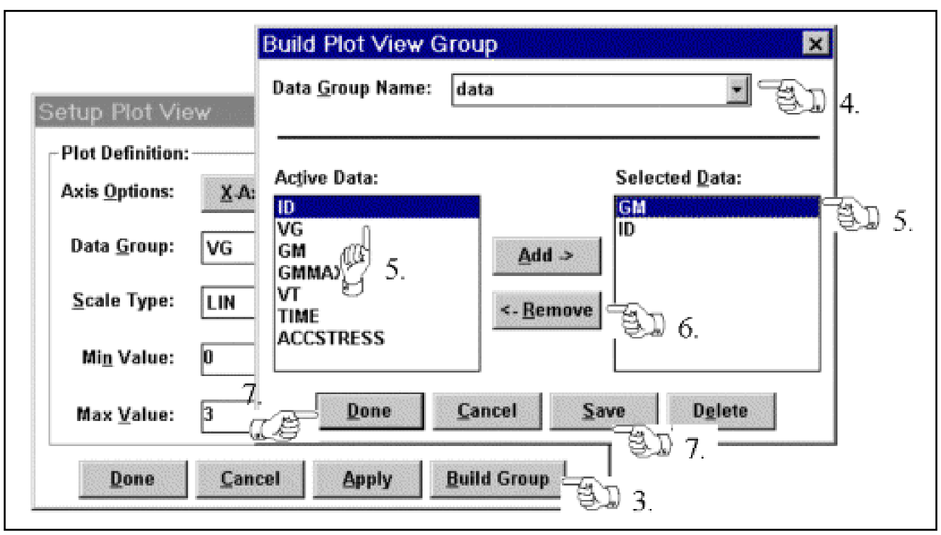


Figure 6-4: How to Edit or Delete a Plot Data Group.

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 to 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, ICS 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, ICS 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, Y_1, and Y_2. 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 Y_1 or Y_2 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 Y_1 or Y_2 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 Y_1-Y_2-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 ICS is shut off and later re-started.

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.

* 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.

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 remained 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.
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 Selec-tion 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:

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

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

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

Minimizing the value of X^2 is the criteria for establishing the linear equation that best fits the data. Notice that X^2 is a function of two unknowns, *a* and *b*. X^2 is minimized by reducing the function to a system of two equations with two unknowns. The first equation is the derivative of X^2 with respect to *a*, and the second equation is the derivative of X^2 with respect to *b*. Both equations are set equal to zero because the minimum value of each derivative is desired. Therefore, the system of two equations and two unknowns is characterized by the two functions:

$$\frac{d\left(x^2\right)}{da} =$$

and

$$\frac{d\left(x^2\right)}{da} =$$

The values of *a* and *b* are determined by simultaneously solving the pair of equations described by the above functions. The solution to the linear curve fit is a line defined by y(x) = a + bx.

Therefore, the slope and y-intercept are taken directly from the derived solution; that is:
a = y-intercept, and
b = slope.
The x-intercept is found by setting y(x) = 0:
0 = y(x) = a + bx

$$x = -\left(\frac{a}{b}\right)$$

The following specifications characterize the fitted line: slope, Y_{int}, X_{int}, error, and the number of data points used in the calculation. The slope, X_{int}, and Y_{int} are reported in the respective Curve Fit Table.

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:

$$y_1(x) = ae^{bx1}$$

and

$$y_2(x) = ae^{bx2}$$

Therefore, finding the exponential solution that satisfies both points is a problem of determining the values of *a* and *b*. ICS 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*.

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

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. ICS 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:

$$y_{tan}(x) = m_{tan}\, x + d$$

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):

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

The general equation of the tangent line now becomes:

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

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:

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

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).

1. Constant Output
   Value
2. Step Output
   Start
   Stop
   Step
   # Pnts
3. 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. Y_1-Axis Label
4. Y_2-Axis Label
5. Comment Line

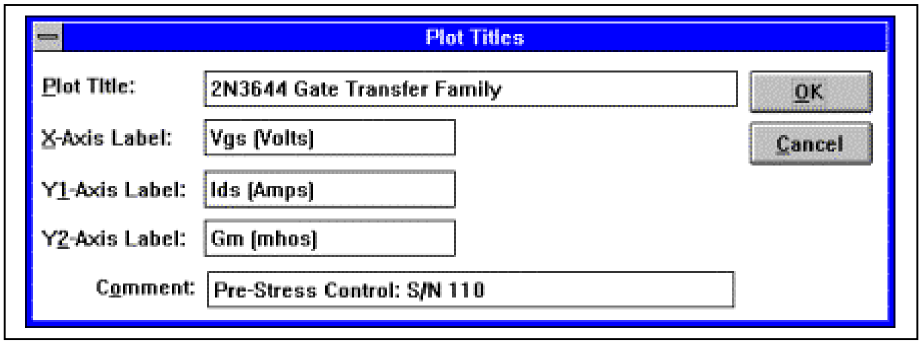


Figure 6-5: The Plot Titles Dialogue Box

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 (Y_2-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 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 ICS. The color composition of the active plot window is controlled from the Plot Colors dialogue box.

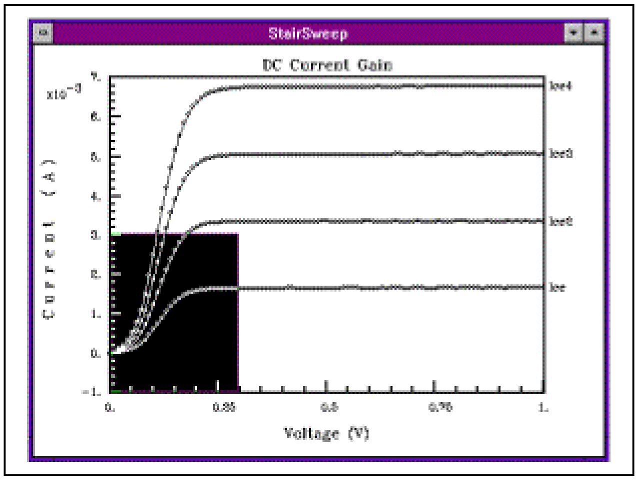
Zoom In

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.

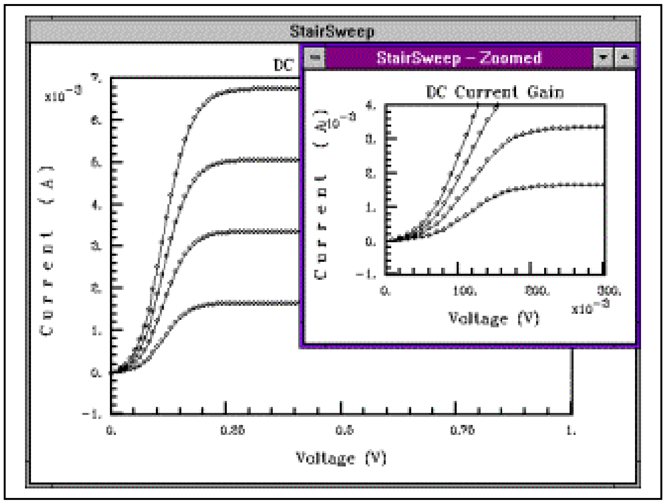
How to Create a Plot Window with the Zoom Feature:

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.

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.

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

How to Store Plots for Overlaying:

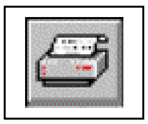
1. Make certain that the plot to be stored for overlay is the active plot window. If it's not, click anywhere in the plot or click twice on the plot window icon.
2. Activate the memory feature by clicking the toolbar MEMORY button. Then from the four memory buttons select the desired memory location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

Chapter 7: Printing, Exporting, and Linking

What can Interactive Characterization Software Do?

Interactive Characterization Software (ICS) supports a full range of printing, exporting, database searching/exporting, and DDE linking 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. ICS also takes full advantage of the Dynamic Data Exchange (DDE) capability available in Windows, allowing data window spreadsheets and plot windows to be dynamically linked to other Windows applications that support DDE.

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:

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

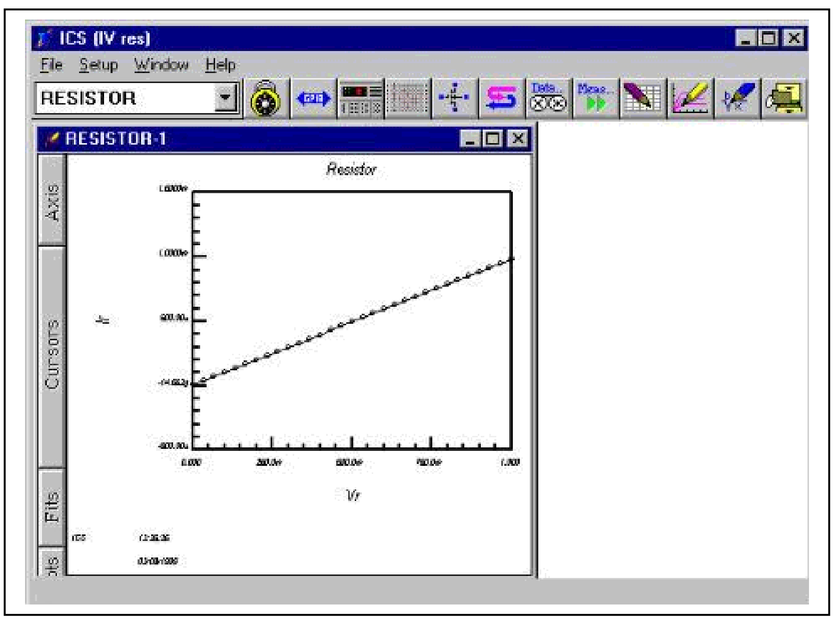
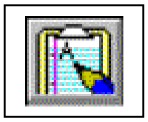


Figure 7-1: How to Print a Data Window Spreadsheet or a Plot Window.

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.

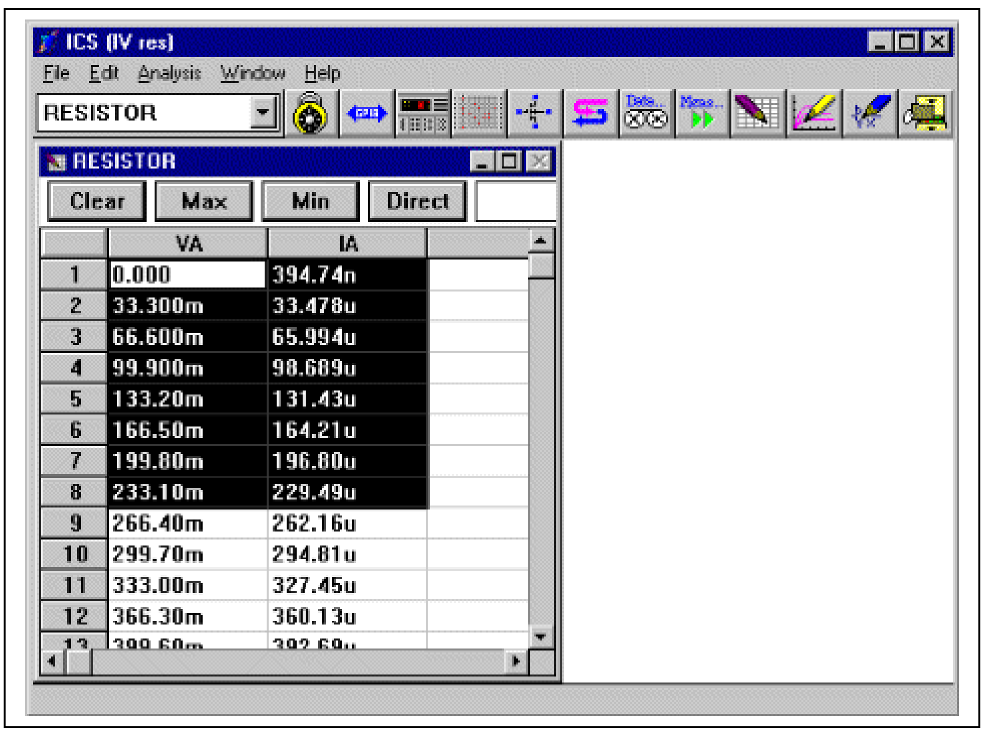


Figure 7-2: How to Copy a Data or Plot Window to the Clipboard.

Sharing Data Between ICS Workstations

ICS allows the sharing of data between ICS 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 ICS 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.

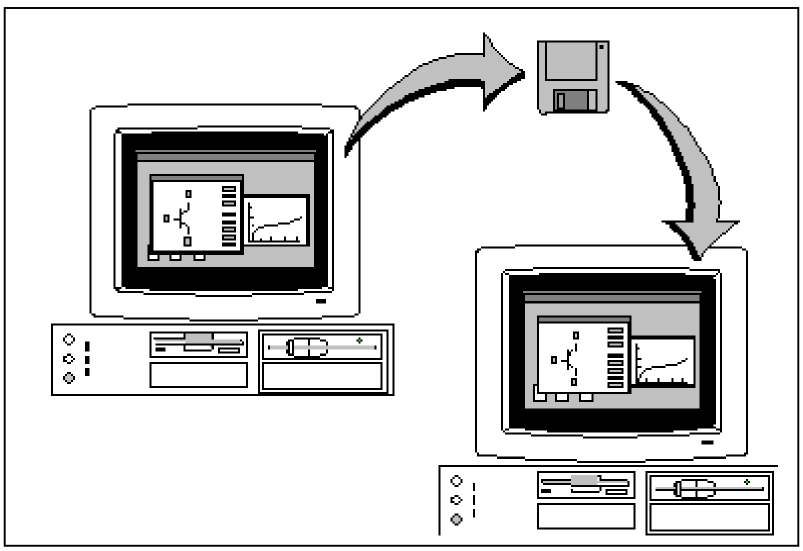


Figure 7-3: The Binary Export Function Provides the Capability to Copy Project Files from one Machine to Another.

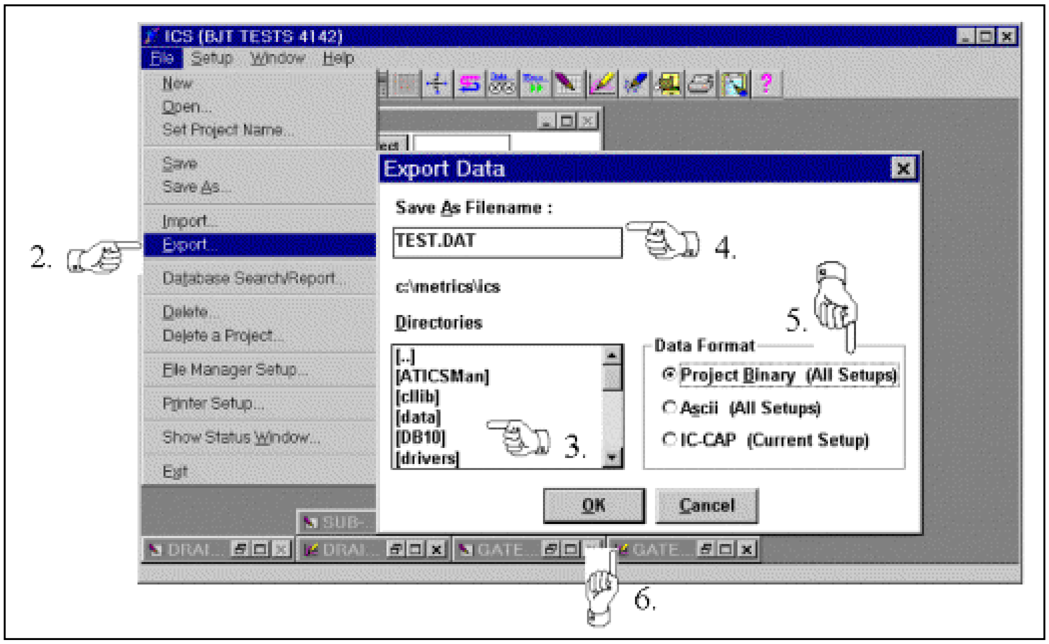


Figure 7-4: How to Export a Project File in Binary Format

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 ICS 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.

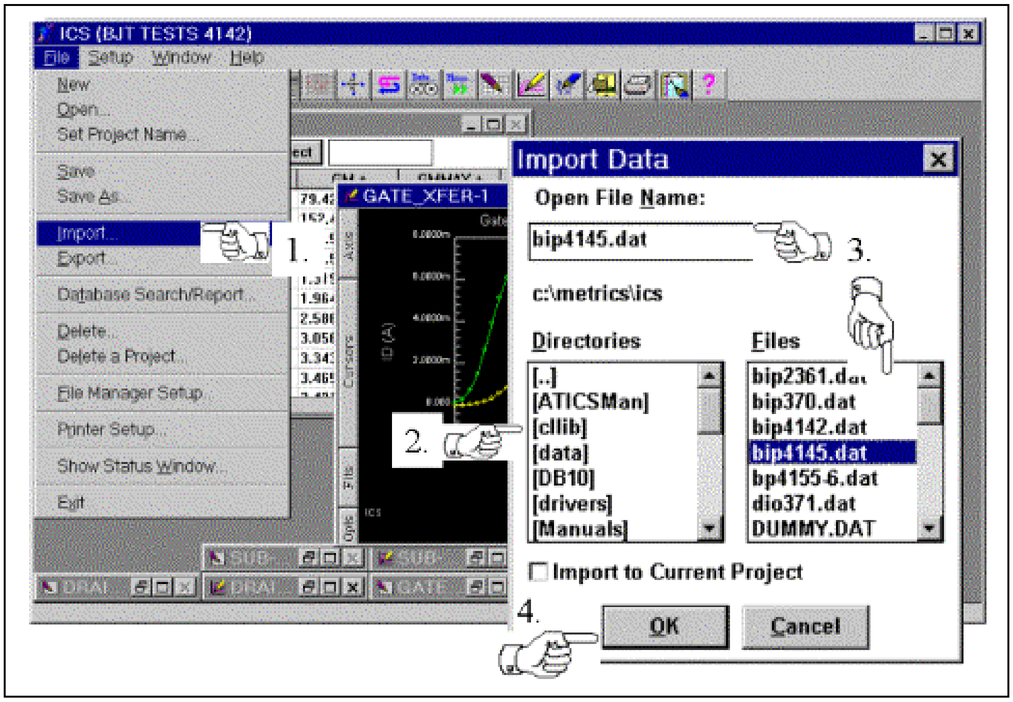


Figure 7-5: How to Import a Project File

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.

* When a project file is opened, the necessary instrument drivers are automatically connected to ICS. Any of the defined test setups can be executed immediately. Thus, it is important to make sure when transporting project files between two systems that both systems have the same drivers available.

Extracting ASCII Data From ICS

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.

ASCII Export of the Current Project

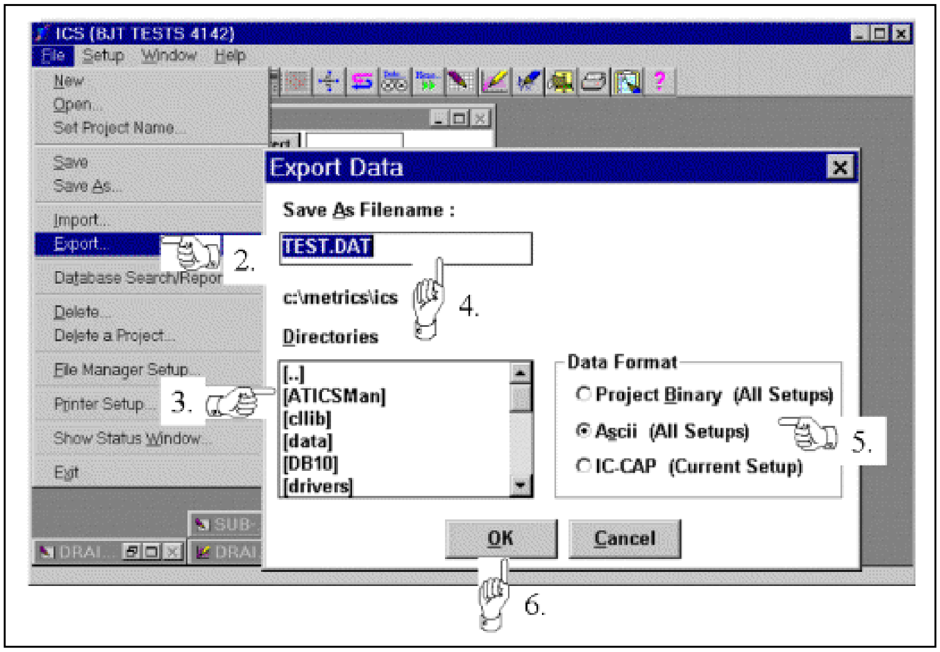


Figure 7-6: How to Export a Project File in ASCII Format

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 ICS has completed exporting the file.

* If you export in binary format, you will not be able to read the file in an ASCII text editor.

Automatic ASCII Logging of Project Data

Automatic ASCII logging of project files generate a file with contents that are identical to files generated with the ASCII Export function, except that with this automatically appends data to an ASCII LOG file after each measurement execution is completed. This process continues until the Enable Logging feature is turned OFF.

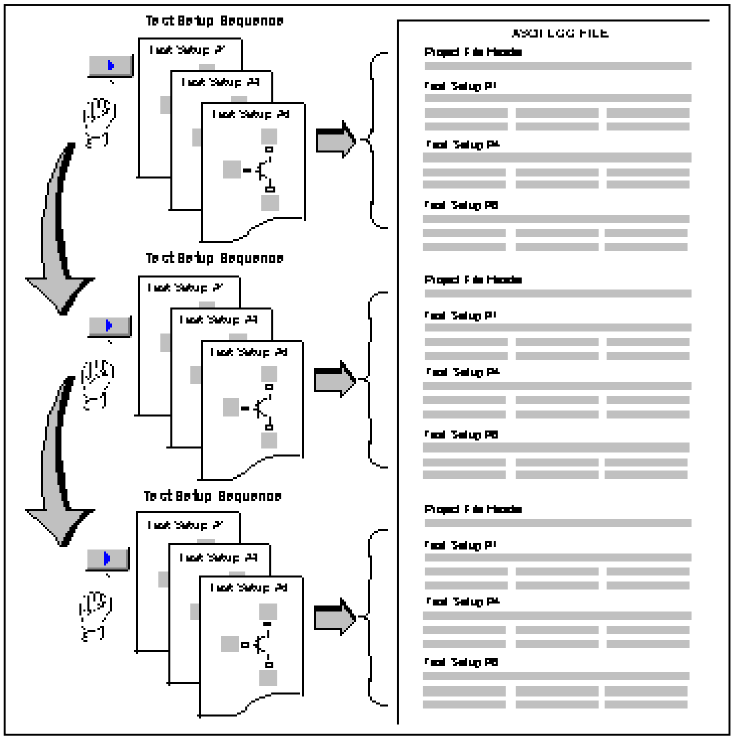
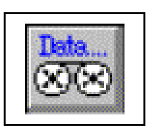


Figure 7-7: An Overview of Automatic ASCII Logging of Data.



The Auto-Export controls are located in the Global Control dialogue box. The Global Control dialogue box is opened by selecting the GLOBAL CONTROL toolbar button. After a measurement sequence is defined and enabled, the Automatic ASCII Logging feature is turned ON by selecting the Enable Logging switch and designating the name and path of the Log file.

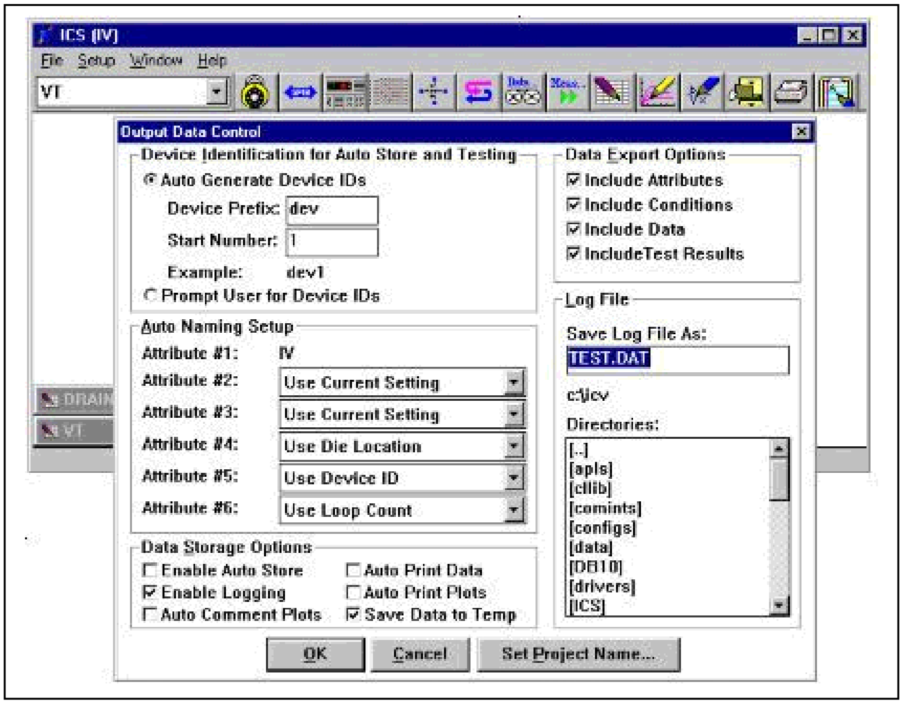


Figure 7-8: How to Configure ICS in Auto-Export Mode.

How to Enable Automatic ASCII Logging of Data:

1. The Auto-Export mode is controlled from the Global Control dialogue box. Open the Global Control dialogue box by clicking the corresponding toolbar button.

7. Select the "Enable Logging" switch to configure ICS in Automatic ASCII Logging mode.

2. As a default, the LOG-file will be named "TEST.DAT" and will be located in the ICS root directory: "C:\ICS". If desired, designate a new log file name in the "Save LogFile As" field and designate a new path from the "Directories" window. To designate a new path, double-click on the available directory name(s) listed in the "Directories" window.

3. Designate what information is to be included in the ASCII log file by enabling the desired items in the Data Export Options.

4. The file attributes window will display the attributes of the current project file. These are the designations that will comprise the project file headers written to the ASCII log file. If the present attributes are unacceptable, specify a new set of attributes by clicking the SET PROJECT NAME button. This will open the File Manager dialogue box. Designate the desired attribute specifications in each of the attribute fields.

5. **NOTE:** Changing the attributes as described in this step WILL change the attribute designations corresponding to the current configuration of the File Manager. If the desktop configuration was already saved to a project file, then selecting FILE/SAVE will actually initiate the FILE/SAVE AS function (assuming the new project file attributes defined in this step do not correspond to an existing project file). If a new project file is opened, the attributes designated in the Global Control dialogue box will be overwritten with the attributes associated with the new project file.

6. If you want to have the user prompted for a device identification or have the device identification automatically generated then configure the "Device Identification for Auto Store and Testing" options. The device ID will automatically be used as the Attribute which is designated to "Use Device ID" in the "Auto Naming Setup".

7. Designate the automatic use of the file manager attributes in the "Auto Naming Setup" controls.
8. Close the Global Control dialogue box by selecting the OK button. ICS will now measure in Auto-Export mode.

Increased Measurement Time

When ICS is configured in Auto-Export mode, ICS will execute the ASCII Export function after each measurement execution. Using the Auto-Export function will increase the time required for each measurement, since ICS will not restore control to the user until ICS is finished writing to the log file.

Extracting IC-CAP Data From ICS

Files exported as IC-CAP data are readable text files that may be used as "SET" files with IC-CAP. IC-CAP "SET" files are files which include source conditions and measured data which may be read into IC-CAP and then used for modeling. Only the current or active setup may be exported as an IC-CAP file.

Export of the Current Setup for IC-CAP

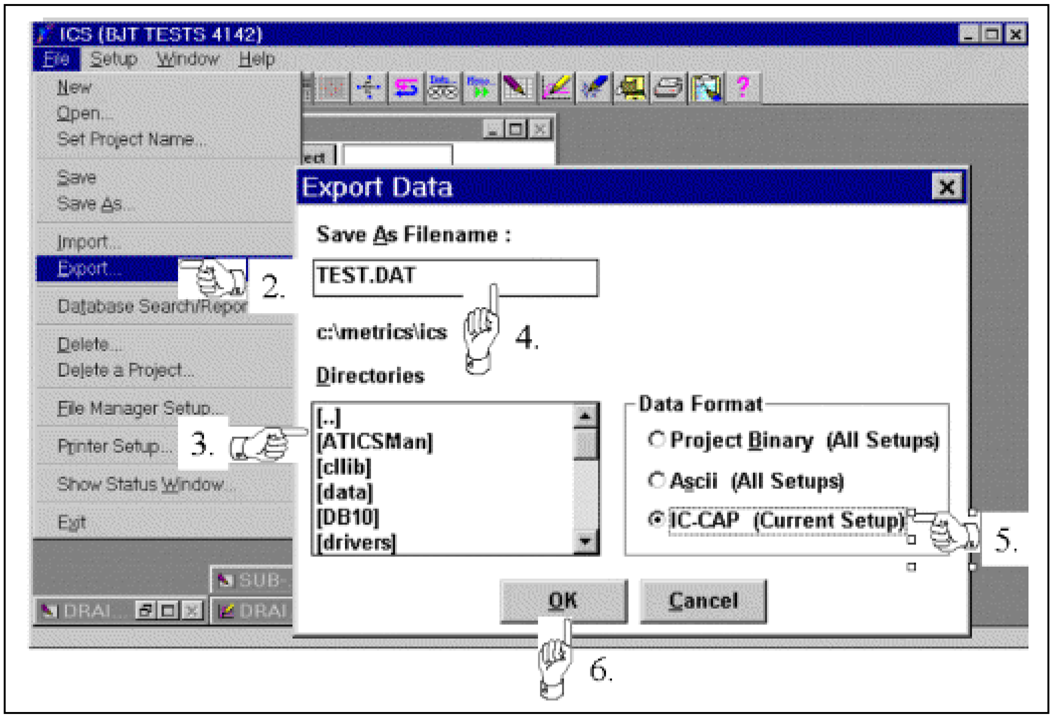


Figure 7-9: How to Export a Setup File in IC-CAP Format

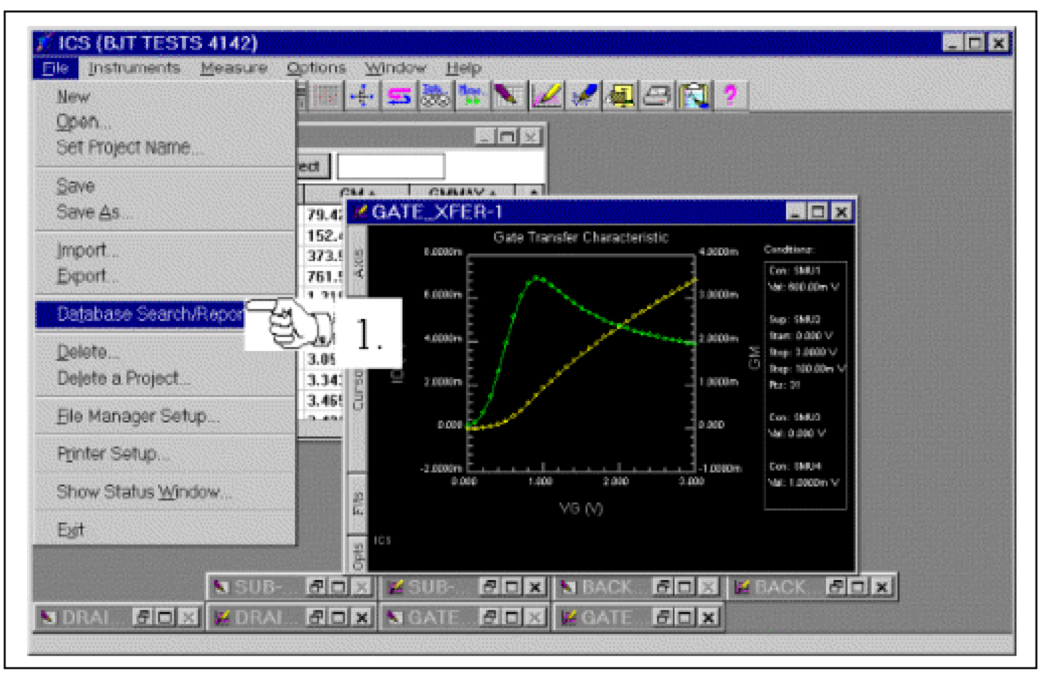
How to Export a Setup File in IC-CAP Format

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 with a "SET" extension for the file you are exporting.
5. Select the IC-CAP switch. **Note:** ICS will only export the current setup in IC-CAP format. Multiple exports must be performed to export multiple setups in IC-CAP format.
6. Click OK. The Export Data dialogue box will close automatically after ICS has completed exporting the file.

Database Searching and Data Manipulation

ICS includes a database search engine which allows a user to search for desired project files through information included in a project files attributes and comments. After acquiring a list of desired project files the user may then perform several functions which include opening a project file, bulk deleting of project files, bulk exporting of project data, and bulk exporting of project test results.

Performing Database Searches



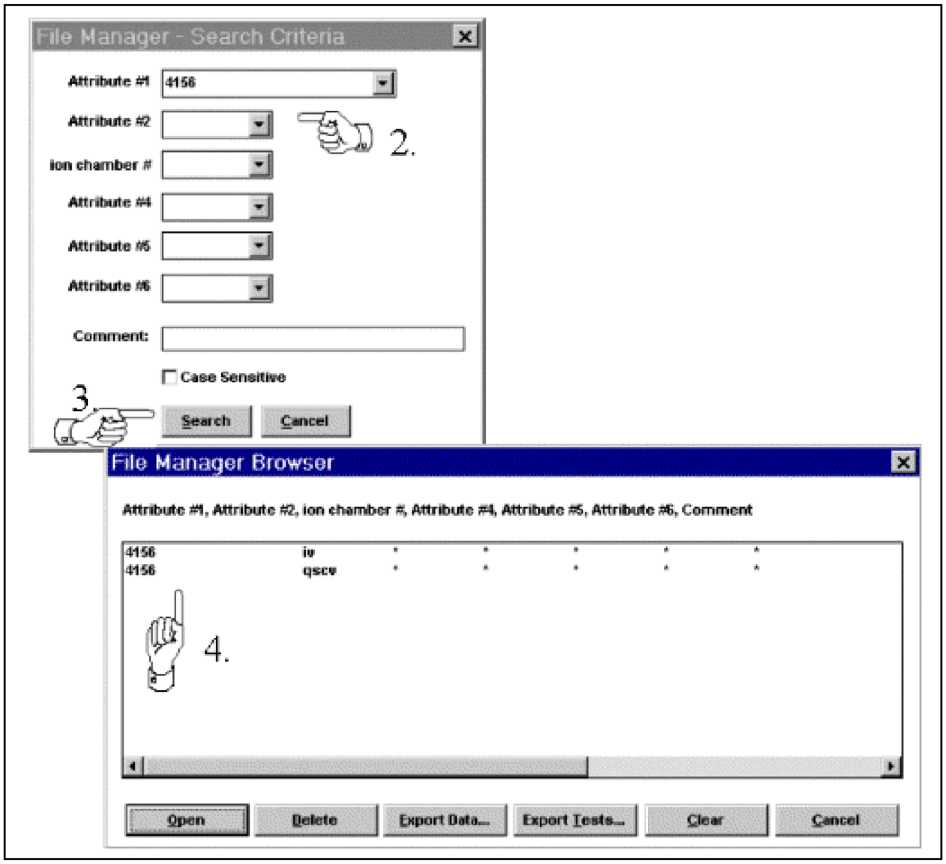


Figure 7-10: Searching for Projects in the ICS Database.

How to Search for Projects in the ICS Database:

1. Select the File|Database Search/Report menu items to display the File Manager Search Criteria dialog box.
2. Enter the desired search criteria strings in the corresponding attribute and comment fields. Clear all attribute and comment fields to display all of the projects contained in the ICS database. Incomplete or partial strings may be used for search criteria.
3. Press the Search button to begin the search.
4. If the records containing the search criteria are found the File Manager Browser dialog box will be displayed and contain the list of projects.

Opening a Project Found in a Database Search

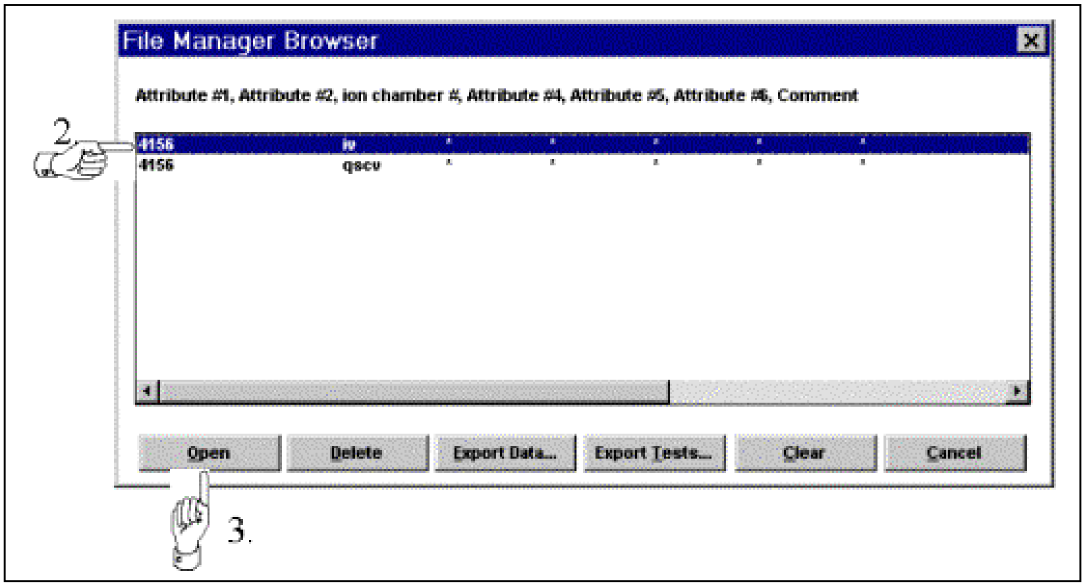


Figure 7-11: Opening a Project in the ICS Database.

How to Open a Projects from a Database Search:

1. Perform the desired database search. See Performing Database Searches section for a full description.
2. Select the project file to be opened from the list of project files shown in the File Manager Browser dialog box.
3. Press the Open button to open the selected project file.

Deleting Project Files Found in a Database Search

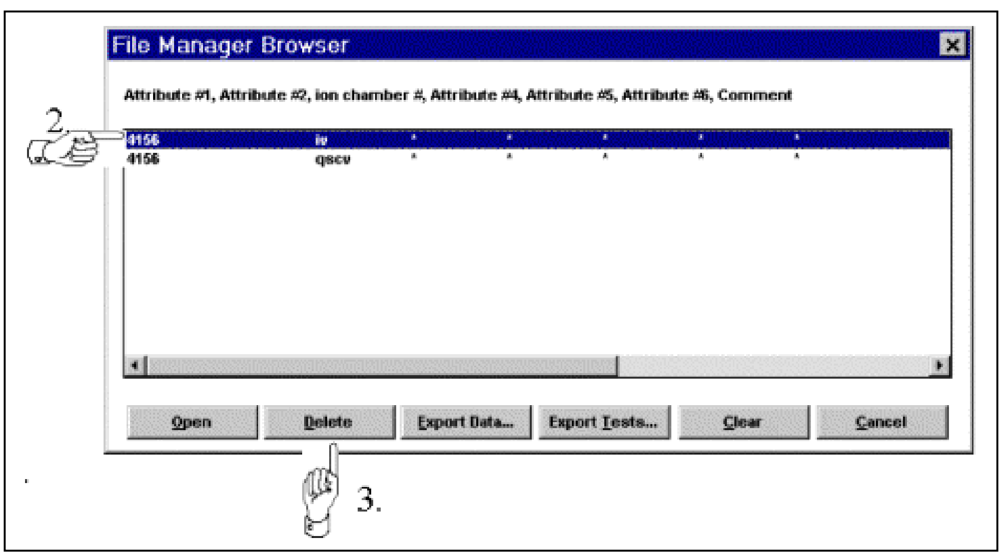


Figure 7-12: Deleting Projects from the ICS Database.

How to Delete Project Files from a Database Search:

1. Perform the desired database search. See Performing Database Searches section for a full description.
2. Select one or more project files to be deleted from the list of project files shown in the File Manager Browser dialog box.
3. Press the Delete button to delete the selected project files.

Exporting Project Files Found in a Database Search

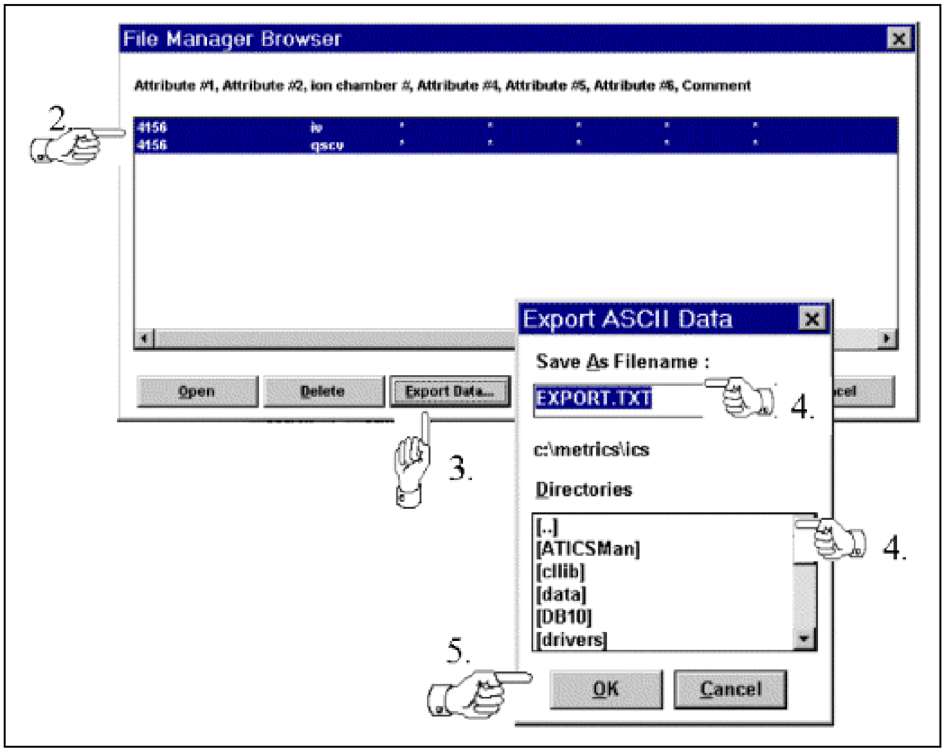


Figure 7-13: Exporting Projects from the ICS Database.

How to Export Project Files from a Database Search:

1. Perform the desired database search. See Performing Database Searches section for a full description.
2. Select one or more project files to be ASCII exported from the list of project files shown in the File Manager Browser dialog box.
3. Press the Export Data button to display the Export ASCII Data dialog box.
4. As a default, the export file will be named "EXPORT.TXT" and will be located in the ICS root directory: "C:\ICS". If desired, designate a new export file name in the "Save As Filename" field and designate a new path from the "Directories" window. To designate a new path, double-click on the available directory name(s) listed in the "Directories" window.
5. Press the OK button to begin exporting the selected project files.

Exporting Project Test Results from a Database Search

Setup test results may be exported in a tabular form that may be imported directly into spreadsheet programs.

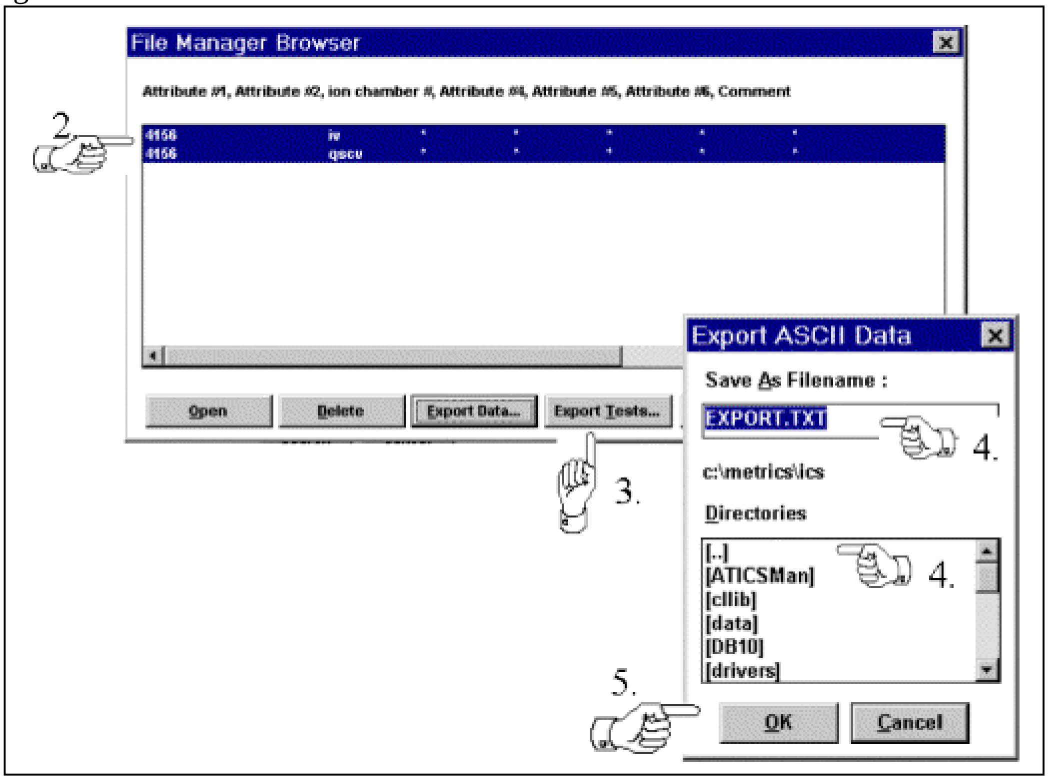


Figure 7-14: Exporting Test Results from the ICS Database.

How to Export Project Test Results from a Database Search:

1. Perform the desired database search. See Performing Database Searches section for a full description.
2. Select one or more project files which contain the test results to be exported from the list of project files shown in the File Manager Browser dialog box.
3. Press the Export Tests button to display the Export ASCII Data dialog box.
4. As a default, the export file will be named "EXPORT.TXT" and will be located in the ICS root directory: "C:\ICS". If desired, designate a new export file name in the "Save As Filename" field and designate a new path from the "Directories" window. To designate a new path, double-click on the available directory name(s) listed in the "Directories" window.
5. Press the OK button to begin exporting the selected project test results.

Dynamic Data Exchange (DDE)

Sharing Data

ICS fully supports the Dynamic Data Exchange capability available in Windows. Data window spreadsheets or plot windows opened in ICS can be dynamically linked to any Windows application supporting DDE.

* Setup names should not contain any spaces if the setup is to be the source of a DDE link.

Creating a Dynamic Link with Excel

Any ICS data window spreadsheet can be linked to an Excel spreadsheet. When changes are made to the data window spreadsheet in ICS, the Excel spreadsheet can be updated by selecting the Excel LINKS/UPDATE command. Once the data is in Excel, all of Excel's worksheet and chart commands can be used to manipulate the data.
The link between ICS and Excel is established through the data window spreadsheet, not the corresponding plot window. Changes made to the ICS plot window will not affect the data in Excel.

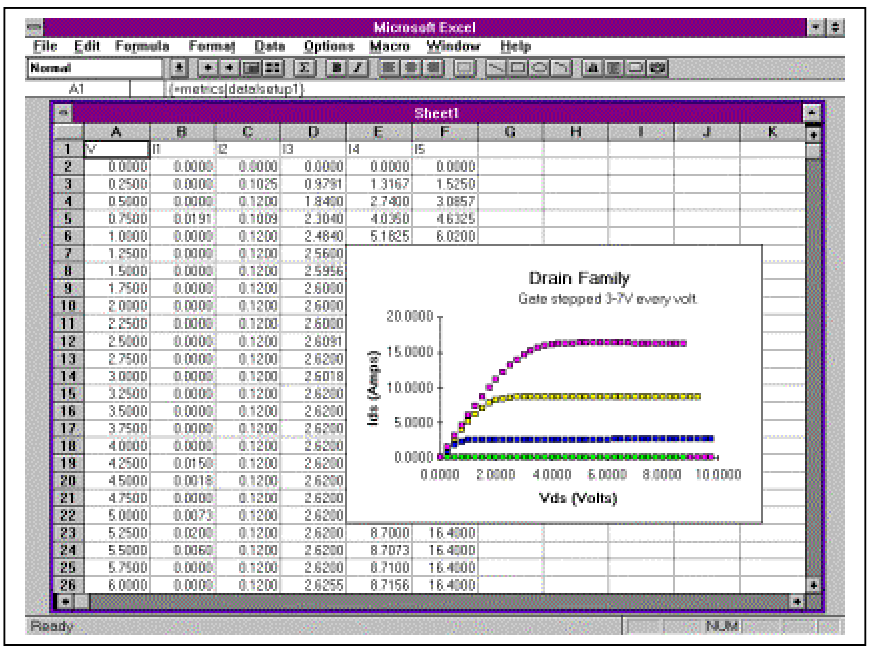
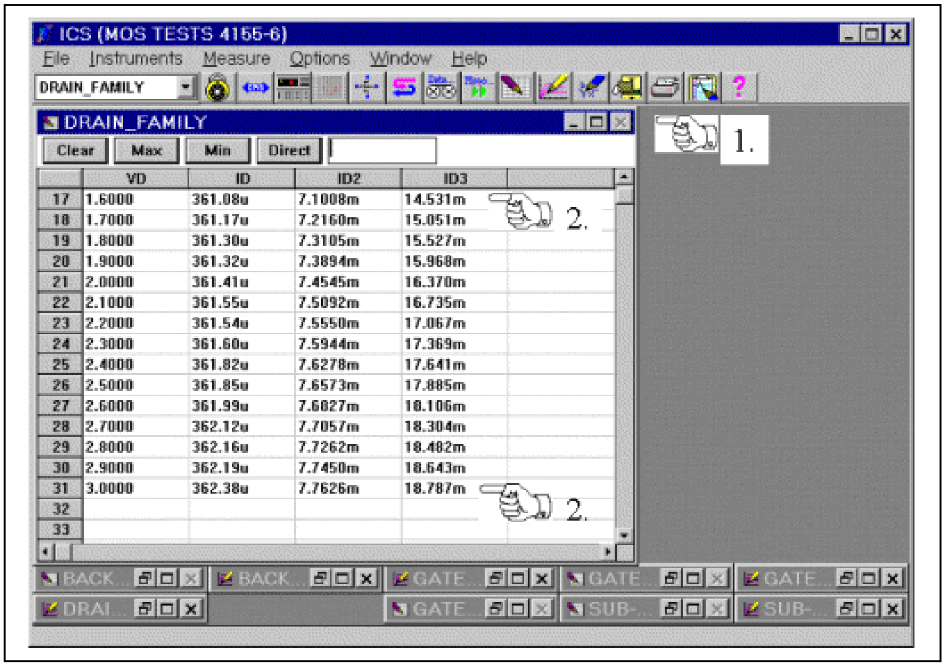
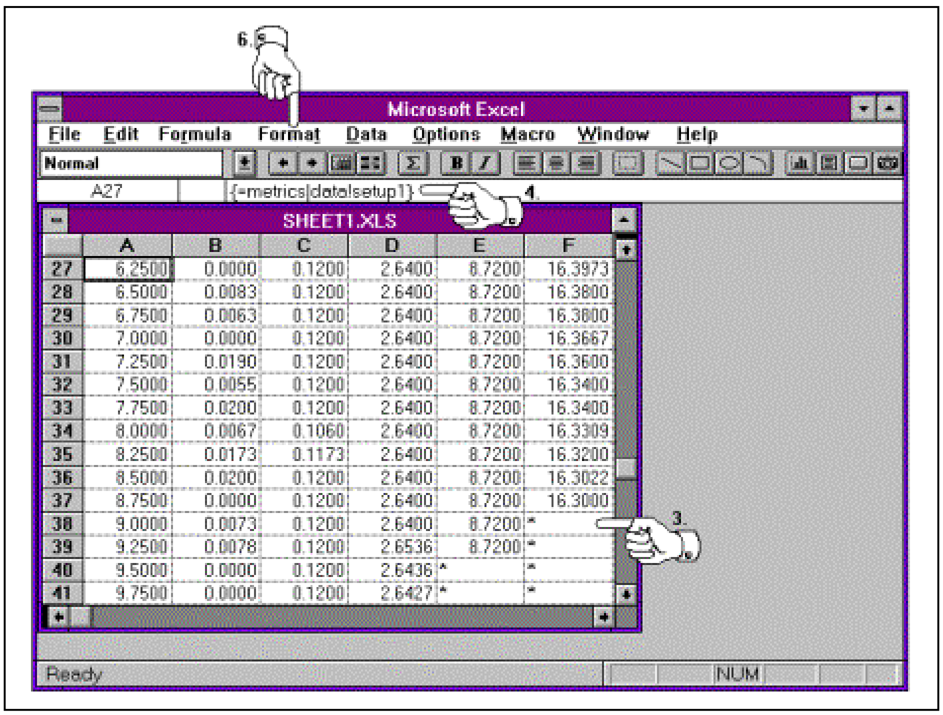


Figure 7-15: Data Linked to Excel from ICS through DDE.



	VD	ID	ID2	ID3
17	1.6000	361.08u	7.1008m	14.531m
18	1.7000	361.17u	7.2160m	15.051m
19	1.8000	361.30u	7.3105m	15.527m
20	1.9000	361.32u	7.3894m	15.968m
21	2.0000	361.41u	7.4545m	16.370m
22	2.1000	361.55u	7.5092m	16.735m
23	2.2000	361.54u	7.5550m	17.067m
24	2.3000	361.60u	7.5944m	17.369m
25	2.4000	361.82u	7.6278m	17.641m
26	2.5000	361.85u	7.6573m	17.885m
27	2.6000	361.99u	7.6827m	18.106m
28	2.7000	362.12u	7.7057m	18.304m
29	2.8000	362.16u	7.7262m	18.482m
30	2.9000	362.19u	7.7450m	18.643m
31	3.0000	362.38u	7.7626m	18.787m
32				
33				

Figure 7-16: How to Create a Link Between ICS and Excel (Part A)



	A	B	C	D	E	F
27	6.2500	0.0000	0.1200	2.6400	8.7200	16.3973
28	6.5000	0.0083	0.1200	2.6400	8.7200	16.3800
29	6.7500	0.0063	0.1200	2.6400	8.7200	16.3800
30	7.0000	0.0000	0.1200	2.6400	8.7200	16.3667
31	7.2500	0.0190	0.1200	2.6400	8.7200	16.3600
32	7.5000	0.0055	0.1200	2.6400	8.7200	16.3400
33	7.7500	0.0200	0.1200	2.6400	8.7200	16.3400
34	8.0000	0.0067	0.1060	2.6400	8.7200	16.3309
35	8.2500	0.0173	0.1173	2.6400	8.7200	16.3200
36	8.5000	0.0200	0.1200	2.6400	8.7200	16.3022
37	8.7500	0.0000	0.1200	2.6400	8.7200	16.3000
38	9.0000	0.0073	0.1200	2.6400	8.7200	*
39	9.2500	0.0078	0.1200	2.6536	8.7200	*
40	9.5000	0.0000	0.1200	2.6436	*	*
41	9.7500	0.0000	0.1200	2.6427	*	*

Figure 7-17: How to Create a Link Between ICS and Excel (Part B)

How to Create a Link Between ICS and Excel:

1. From ICS, designate the desired test setup as the active setup. Open the corresponding data window spreadsheet.
2. Determine the data window spreadsheet array size, for example, 4 columns by 30 rows.
3. In Excel, highlight the appropriate number of columns and rows that correspond to the array size of the data window spreadsheet.
4. At the Excel command line, enter:

    =ics|data!*SetupName*

    where:

    - "*SetupName*" is the name of the test setup as it appears underneath the corresponding data window spreadsheet. A list of available test setups can be displayed by clicking on the toolbar Setup window scroll box.
    - "|" is a "pipe". The "pipe" is a keyboard key usually shown as two vertical dashes on top of each other. On most keyboards the "pipe" symbol is usually on the same key as the backslash (\).
    - "!" is an exclamation point.
5. After entering the Excel syntax, simultaneously hit Shift Ctrl Enter.
6. After Excel reads and displays the data, the data format may not be standardized. Standardize the presentation of the data by selecting FORMAT/NUMBER from the Excel worksheet menu.