Metrics ICS Driver Manual

Tektronix 370/370A/370B

Metrics ICS

Version 4.5

Table of Contents

Getting Started: Creating and Executing a Test Setup

This section will walk you through the steps required to create and execute a test setup that measures diode turn-on voltage using the Tek370 Programmable Curve Tracer. This is simple measurement, but it will provide you with a general understanding of how ICS and the Tek370 are used to measure device characteristics.

Step 1: Connect the Tek370 Instrument Driver

Step 2: Specify the GPIB Address

Step 3: Create the Test Setup

Step 3A: Specify the Test Setup Name

Step 3B: Select a Device Schematic Corresponding to the DUT

Step 3C: Designate the Supply/DUT Connections

Step 3D: Specify the Source/Measure Configuration of Each Supply

Step 4: Insert the DUT Into the Test Fixture

Step 5: Execute the Measurement

Step 6: View the Results

Step 7: Create a Plot of the Results

Step 8: Save the Results into a Project File

Step 1: Connect the Tek370 Instrument Driver

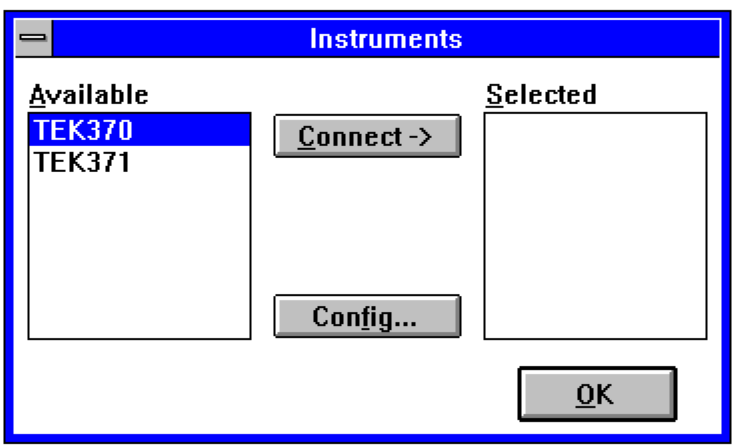
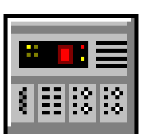


Figure 1: Instrument driver connection dialog box.

The Tek370 Driver is connected in the Connect Instruments dialog box. The Connect Instruments dialog box is accessed by clicking the toolbar CONNECT INSTRUMENTS button or by choosing INSTRUMENTS\SELECT INSTRUMENT from the measurement mode menu bar.



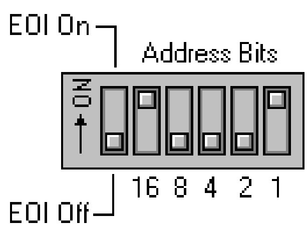
Click the toolbar CONNECT INSTRUMENTS button to display the Connect Instruments dialog box.

How to Connect the Tek370 Instrument Driver:

1. Click the CONNECT INSTRUMENTS toolbar button or select INSTRUMENTS/SELECT INSTRUMENT from the measurement mode menu bar. This will open the Connect Instruments dialog box.
2. Highlight the Tek370 Driver in the AVAILABLE field. If the Tek370 Driver does not appear in the AVAILABLE field, verify that the TEK370.DLL file is present in the C:\ICS\DRIVERS directory
3. Click the CONNECT button.
4. Your choice will be displayed in the SELECTED field.
5. Clicking the OK button would close the Connect Instruments dialog box and restore control to the ICS desktop. Keep the Connect Instruments dialog box displayed for now, because the next step requires you to click the Connect Instruments CONFIG button.

Step 2: Specify the GPIB Address

The Tek370 must be connected to your computer with the use of a standard IEEE-488 Bus. This section provides the basic information necessary to establish the hardware connection between the Tek370 and an 80386/80486 machine. Please refer to section 4-1 of the Tektronix 370 Operation Manual to review the details of GPIB interface control.



Make sure the Tek370 is connected to your computer using a standard IEEE-488 Bus. Before installing the bus, make certain that the rear-panel GPIB Address switches are configured correctly. The EOI Bit must be set to the OFF position. The remaining five bits are the address bits and can be configured to any binary value between 0 and 30.

The Tek370 Configuration dialog box is used to specify the GPIB address configured on the instrument's rear-panel GPIB Switch. The ID and ROM displays are static display fields only and do not require any definition by the user. The content of the ID and ROM displays are determined by an instrument poll when the CONFIG button is selected from the Connect Instruments dialog box.

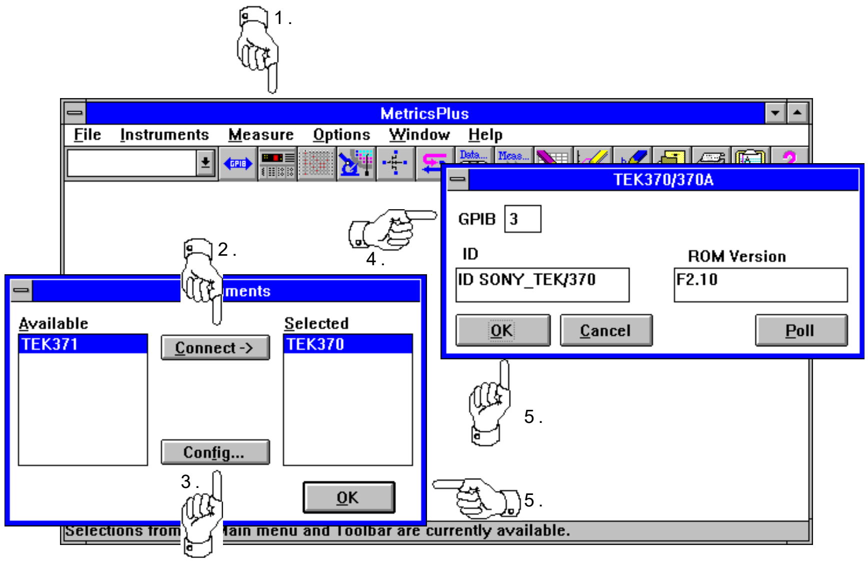


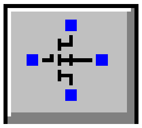
Figure 2: How to Specify the Tek370 GPIB Address

How to Specify the Tek370 GPIB Address:

1. If the Connect Instruments dialog box is no longer displayed, click the toolbar CONNECT INSTRUMENTS button or choose INSTRUMENTS/SELECT INSTRUMENT from the measurement mode menu bar.
2. Make certain that the Tek370 Driver is connected. If not, highlight the desired driver in the AVAILABLE field and click the CONNECT button. Refer to *Step 1: Connect the Tek370 Driver* for more information regarding the connection procedure.
3. Open the Tek370 Configuration dialog box by clicking on the CONFIG button at the bottom of the Connect Instruments dialog box. Selecting the CONFIG button will also poll the instrument to determine the instrument identity and ROM version.
4. Enter the Tek370 GPIB address in the GPIB field. The GPIB address is determined by evaluating the position of the five binary address bits on the GPIB Address switch located on the back panel of the instrument. The GPIB Address switch specifies a binary address between 0 and 31. If you wish to change the GPIB setting, refer to Section 4-1 of the Tek370 Operation Manual.
5. Click the OK button located at the bottom of the dialog box. This will close the Configuration dialog box. Click the OK button in the upper right-hand corner of the Connect Instrument dialog box to restore control to the ICS desktop.

Step 3: Create the Test Setup

Test setups are created in the Setup Editor. Open the Setup Editor by selecting the SETUP EDITOR toolbar button. In this example, we will create a test setup that measures the forward current of an NP diode with respect to a forward voltage sweep.



Click the corresponding toolbar button to display the Setup Editor.

Step 3A: Specify the Test Setup Name

When creating a new test setup, a test setup name must be specified before any other selections or conditions are designated.

How to Specify the Test Setup Name

1. Click the Setup Editor NEW button. This will open the New Setup dialog box.
2. At the prompt, specify a test setup name. For this example, type "DiodeOn".
3. Click OK. This will close the New Setup dialog box.
4. The test setup name will appear in the Setup Editor SETUP window.

Step 3B: Select a Device Schematic Corresponding to the DUT

A device schematic is located at the center of the Setup Editor. The device schematic is designed to provide a graphic image of the test fixture socket.

A MOSFET schematic will appear at the center of the Setup Editor when the Setup Editor is first opened. In this step the MOSFET schematic will be replaced with a diode schematic. The MOSFET device is the default Setup Editor schematic. To change the default device schematic, refer to *Chapter 2: The Setup Editor* in the ICS Reference Manual.

How to Select a Device Schematic:

1. Click the Setup Editor DEVICE button. This will open the Device dialog box.
2. The Device Type window will display a list of available device schematics. Select DIODE. Notice the selected schematic is previewed in the small window to the right of the Device Type window.
3. Some device schematics will display a set of polarity switches when selected. Select the "NP" designation for this example.
4. Click OK. This will close the Device dialog box and display the diode schematic at the center of the Setup Editor.

Step 3C: Designate the Supply/DUT Connections

The supply/DUT connections are designated in the Setup Editor. The Setup Editor display is provided as a tool to document the orientation of the DUT in the test fixture. The connections designated in the Setup Editor must correspond to the reality of your hardware arrangement.

The Setup Editor displays a device schematic representing the DUT. Connections are designated by first clicking one of the available supplies listed in the Source Units dialog box. After the supply is selected, click the blue pad next to one of the device schematic pins. Select the blue pad corresponding to the DUT pin that the supply will be physically connected to. An instrument icon, along with the name of the connected supply, will appear above the device schematic pin as a means of indicating the connection.

This example will show how to connect the collector supply to the anode of an NP diode and the emitter supply to the cathode. The Setup Editor configuration for this test setup is pictured on the following page.

How to Designate the Supply/DUT Connections:

1. Select the Setup Editor SOURCES button. This will open the Source Units dialog box.
2. The Source Units dialog box will display a list of available supplies, or "source units".
3. Click on the desired supply. For this example, click on the "TEK370.CS" designation.
4. Click on the blue pad next to the desired pin in the device schematic. For this example, designate the TEK370.CS connection by clicking on the blue pad next to the diode anode.
5. Repeat this process for each supply connected to the DUT. Finish designating the supply/DUT connections for the DiodeOn test setup by selecting the "TEK370.ES" label and attaching it to the cathode.
6. After all of the supply/DUT connections are designated, close the Source Units dialog box by double-clicking the "-" in the upper left-hand corner of the dialog box.
7. Supply/DUT connections cannot be disconnected. If an error is made while designating a supply/DUT connection, erase the test setup by clicking the Setup Editor DELETE button and answering YES to the prompt. Repeat Steps 4A through 4C.

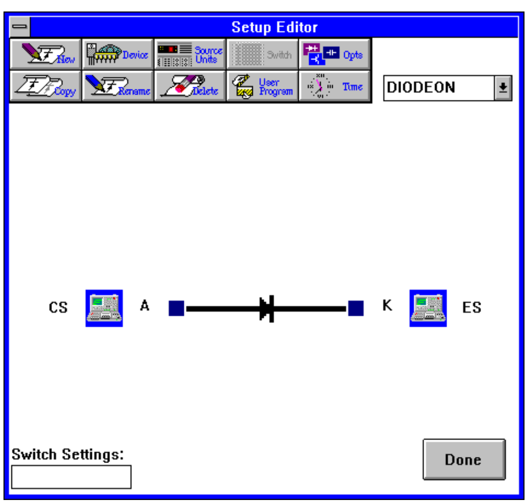


Figure 3: The Supply/DUT Connections Corresponding to the "DiodeOn" Test Setup

Step 3D: Specify the Source/Measure Configuration of Each Supply

Each supply has its own Source Unit Setup dialog box used to specify the source/measure configuration of the supply. Once a supply is connected to a DUT location in the Setup Editor, the corresponding Source Unit Setup dialog box is opened by clicking on the instrument icon displayed above the supply/DUT connection.

In this example, the collector supply (connected to the anode) will force a linear voltage sweep from 0.0V to 1.0V. The sweep will consist of 51 data points and will be bound by a 0.08W power limitation. The collector supply will return both voltage (V) and current (I).

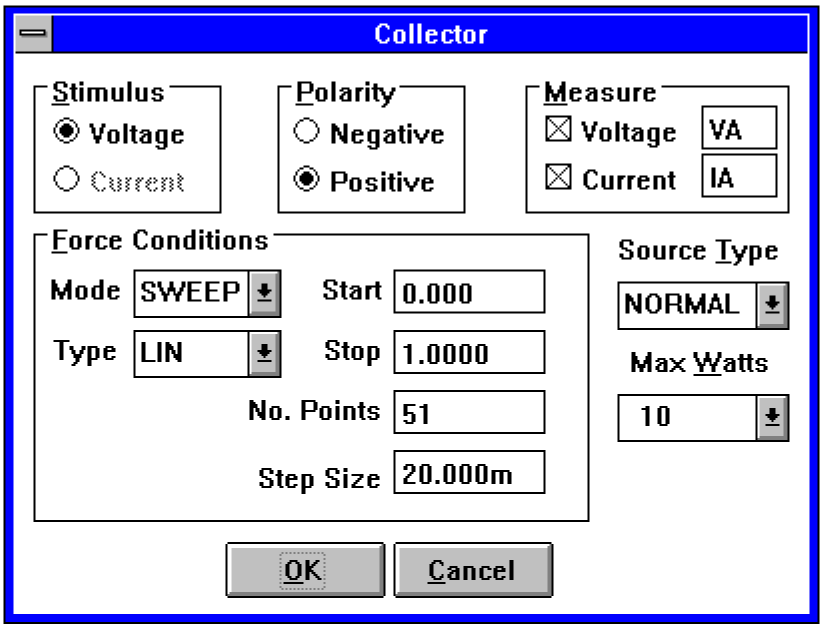


Figure 4: The Collector Supply Source/Measure Specifications for the "DiodeOn" Test Setup.

How to Specify the Source/Measure Configuration of Each Supply:

1. Click once on one of the displayed instrument icons to open the Source Unit Setup dialog box corresponding to the connected supply.
2. Configure the collector supply as shown in Figure 4. Use the mouse or TAB key to move between the different switches and fields in each Source Unit Setup dialog box.
3. Click OK to close a Source Unit Setup dialog box. Only one Source Unit Setup dialog box can be opened at a time.

Step 4: Insert the DUT Into the Test Fixture

Insert the DUT into the test fixture according to the supply/DUT connections designated in the Setup Editor.

Step 5: Execute the Measurement

Execute the DiodeOn test setup by clicking the toolbar MEASURE button. Shortly after the measurement is in process, a message will appear on your screen telling you that the test setup is being executed.

After a few moments another message will be displayed telling you that the collector supply reached compliance. Click IGNORE to complete the measurement.

Step 6: View the Results

Data is automatically generated in the corresponding data window spreadsheet each time the measurement is executed. To display the numerical data, double-click on the white spreadsheet icon labeled "DiodeOn" at the bottom of the ICS desktop. The spreadsheet existed before you executed the measurement, but it contained no data.

Data window spreadsheets are dynamically linked to the test setup. Each time the corresponding test setup is executed, the spreadsheet data is replaced with the most recently measured data. For this reason the data window spreadsheet is automatically named the same as the test setup.

Step 7: Create a Plot of the Results

A plot window is dynamically linked to a corresponding data window spreadsheet. This means that the plot is regenerated any time there is a change to the corresponding spreadsheet data. If the test setup is executed more than once, the plot window is regenerated after each measurement. If the data window spreadsheet is edited, the plot window is updated by clicking the REDRAW button at the top of the spreadsheet. Up to ten plots can be created from a single data window spreadsheet, and each plot can be independently formatted.

The steps below will show you how to create a plot of diode current with respect to the forward voltage sweep. This plot will correspond to the DiodeOn data.

How to Create a Plot

1. If there is more than one defined test setup, designate the active test setup in one of two ways:

   a. Click once on the appropriate data window spreadsheet icon (the data window can be either displayed or minimized).

   b. Click the toolbar setup window arrow and select the desired setup from the displayed drop-down list.

2. Click the NEW PLOT toolbar button. This will open an empty plot window and the Plot Data dialog box.
3. Designate the independent variable of the plot by double-clicking on the appropriate data vector listed in the Data window. Only two quantities were measured in the DiodeOn test setup, voltage and current. There should be two data vectors in the dialog box Data window: "V" and "I". This example will create a plot of current with respect to voltage. Since voltage will be the independent variable, double-click on "V". Notice that the X-button is now labeled with a "V".

4. Designate the first dependent variable of the plot (in our case the only dependent variable) by double-clicking the appropriate data vector in the dialog box Data window. For this example, double-click on the "I". Notice that the Y1-button is now labeled with an "I".
5. You could plot up to nine more data vectors with respect to the independent variable if more data vectors were measured. You can measure more than ten data vectors, but only ten data vectors can be plotted in a single plot window.
6. Click the dialog box APPLY button. This will create the plot but will not close the Plot Data dialog box. You should notice that at about 0.6V the diode turned on.
7. Click the CLOSE button to close the Plot Data dialog box.

Step 8: Save the Results into a Project File

A project file includes all of the information necessary to execute a test setup or group of test setups. A single project file includes: 1) the instrument driver selection, 2) any defined test setup(s), and 3) all of the data and plot windows associated with the test setup(s). For more information about project files, refer to *Chapter 1: How ICS Stores Information, and The File Manager* in Chapter 3 in the ICS Technical Reference Manual.

How to Save Your Work as a Project File

1. Click the SAVE AS toolbar button or select FILE/SAVE AS... from the menu bar. This will open the Windows File Manager.
2. Enter the filename for the project file with the .ICS file extension.
3. Click OK. You just created your first project file.

If you close ICS and want to open this project file later, follow the steps outlined below.

How to Open the Project File

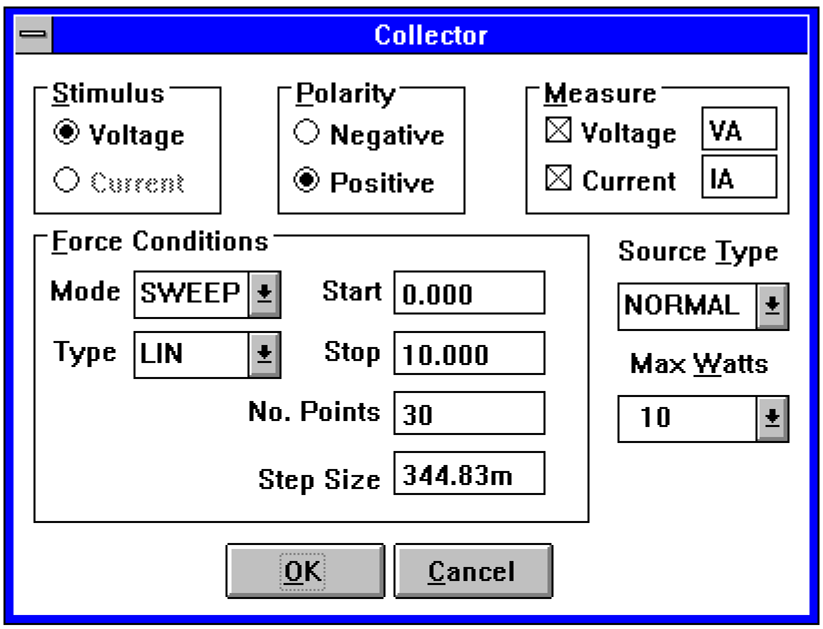
1. Select FILE/OPEN from the ICS menu bar. This will open the Windows File Manager.
2. Select the Project file you want to open by clicking on the .ICS file listed in the File Open dialog.
5. Click OK. This will load the instrument driver(s), test setup(s), and data corresponding to the project file selected.

Using the Tek370 Driver

The Tek370 and Tek370A Programmable Curve Tracers are controlled with the Tek370 Driver. The driver organizes the instrument controls into four dialog boxes: the Collector Supply dialog box, the Base Supply dialog box, the Auxiliary Supply dialog box, and the Tek370 Mainframe Control dialog box.

Common Controls

One of the key advantages provided by the Tek370 Driver is the capability to configure the output of each instrument supply in either a sweep, step, or constant mode. The configuration of either instrument supply is independent of the other. Because of this advantage, each instrument supply dialog box includes a common set of controls. This section describes the controls common to each instrument supply dialog box. The sections that follow describe the controls specific to either the base or collector supply only.



Each instrument supply dialog box includes a set of common controls used to configure the output of each supply in a sweep, step, or constant mode.

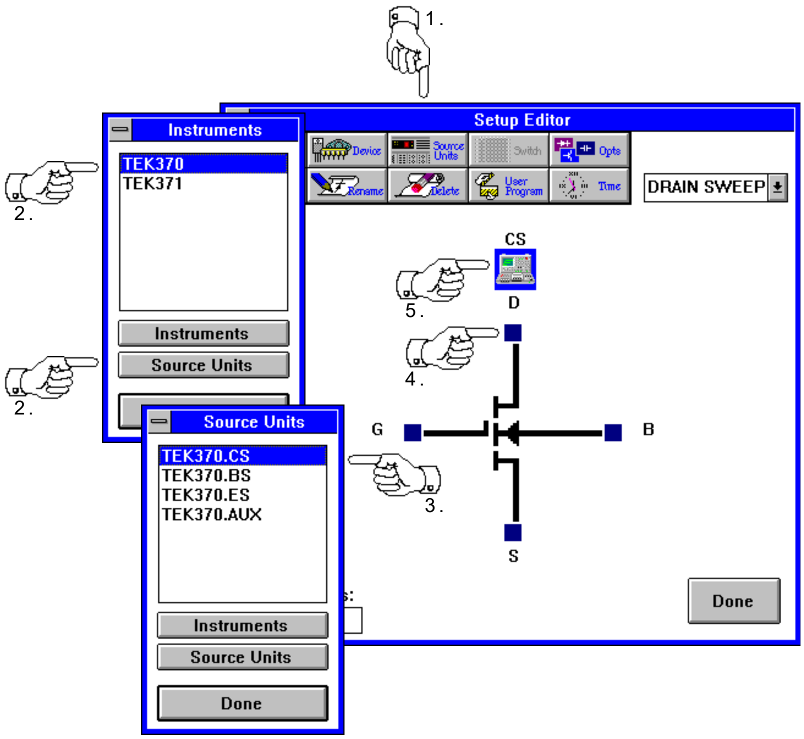


Figure 5: How to Display a Tek370 Instrument Supply Dialog Box

How to Display an Instrument Supply Dialog Box:

1. Open the Source Units dialog box from the Setup Editor.

   a. If additional drivers besides the Tek370 are connected to ICS, the Source Units dialog box will display a list of corresponding instrument designations. Continue with Step #2.

   b. If the Tek370 Driver is the only driver connected to ICS, the Source Units dialog box will display a list of available instrument supplies. Continue with Step #3.

2. Open the menu of available instrument supplies by highlighting the TEK370 designation and clicking the SOURCE UNITS button at the bottom of the dialog box. The source units list can also be displayed by double clicking on the TEK370 designation.
3. Highlight the desired instrument supply.
4. Connect the selected supply to one of the DUT pins displayed in the Setup Editor device schematic by clicking on the respective blue pad. An instrument icon will appear above the connection.
5. Open the instrument supply dialog box by clicking on the instrument icon.

An example of a Tek370 instrument supply dialog box is shown below. The controls common to each Tek370 instrument supply dialog box is identified and described in the following section. For a description of the controls specific to a particular instrument supply, please refer to the sections of this chapter entitled *Specialized Base Supply Controls* or *Specialized Collector Supply Controls*.

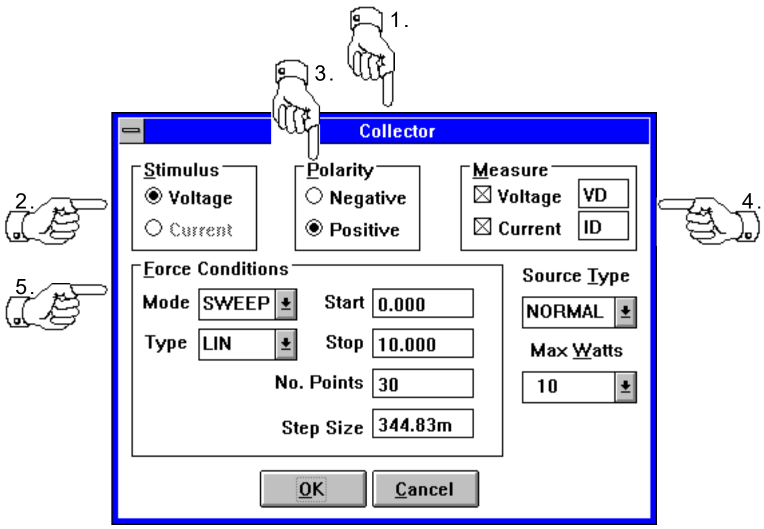
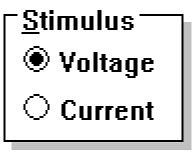


Figure 6: An Example of a Tek370 Instrument Supply Dialog Box

1. **Dialog Box Title Bar**: The identity of the corresponding instrument supply is displayed in the dialog box title bar.
2. **Stimulus Controls**: This switch pair is used to identity whether the forcing signal is a voltage or a current.
3. **Polarity Controls**: This switch pair is used to specify whether the forcing signal is positive or negative.
4. **Measure Controls**: This switch pair is used to specify the type of signal the instrument supply will return as a measurement. This group also includes the text fields necessary to label the returned measurements.
5. **Force Conditions Controls**: The fields included within this group are used to define the shape of the instrument supply forcing signal as well as the forcing signal's start and stop values.

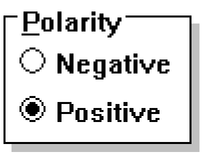
Stimulus Controls



The Stimulus controls consist of two switches: VOLTAGE and CURRENT. These switches are used to specify the characteristic of the forcing signal.

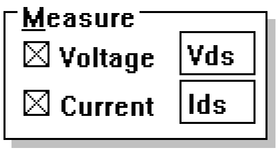
Depending upon the instrument supply, the CURRENT switch may be unavailable. For example, the Tek370 Collector Supply does not include a current source.

Polarity Controls



The Polarity controls consist of two switches: NEGATIVE and POSITIVE. These switches are used to specify the polarity of the forcing signal.

Measure Controls



The Measure controls consist of two switches: VOLTAGE and CURRENT. In addition to the two measure switches, a text field is located to the right of each switch. These switches and text fields are used to specify and label the data that will be returned when the test setup is executed.

An instrument supply's measurement configuration is specified by turning on or off the VOLTAGE and CURRENT switches. Depending upon the particular supply, one of the two switches may be unavailable. For example, the collector supply can measure both voltage and current, while the base supply can measure only current, and the auxiliary supply can measure only voltage.

How to Specify Returned Data:

1. Open the instrument supply dialog box corresponding to the instrument supply that will return data.
2. Click on one or both of the measure switches, depending upon the data you wish to return.
3. After you click on either of the measure switches, the cursor will automatically move to the text field next to the corresponding switch. Specify a variable name for the measured characteristic.
4. Click the OK button to close the dialog box and restore control to the Setup Editor.

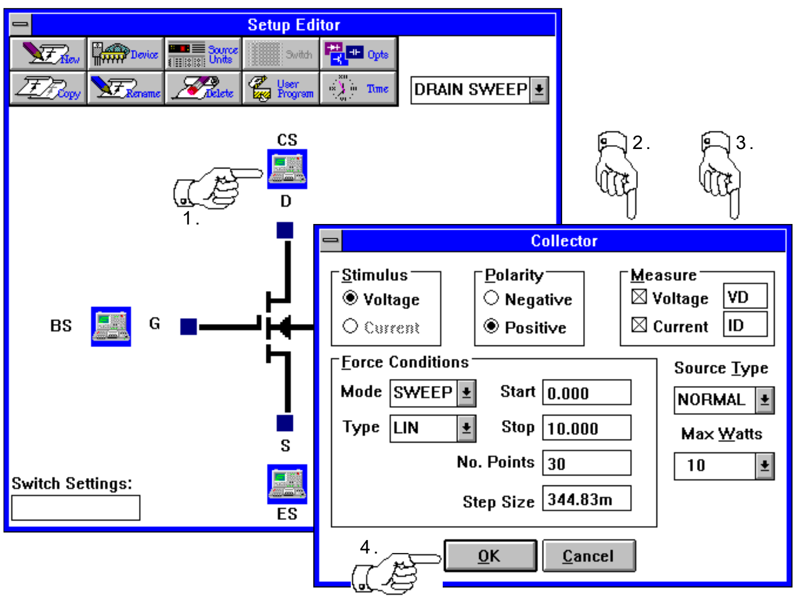


Figure 7: How to Specify Returned Data

Labeling Measured Data

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

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

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

Sequential Data Vectors

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

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

Non-Sequential Data Vectors

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

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

Mode

To select the desired forcing mode, click the scroll arrow in the MODE field. Clicking on the scroll arrow will display the available sweep mode selections. Click on the desired selection.

Force Conditions Controls

The Force Conditions controls are used to specify the shape and boundary of the instrument supply's forcing signal. The Tek370 Driver provides the capability to configure any instrument supply in a sweep, step, or constant output mode.

Sweep Mode

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

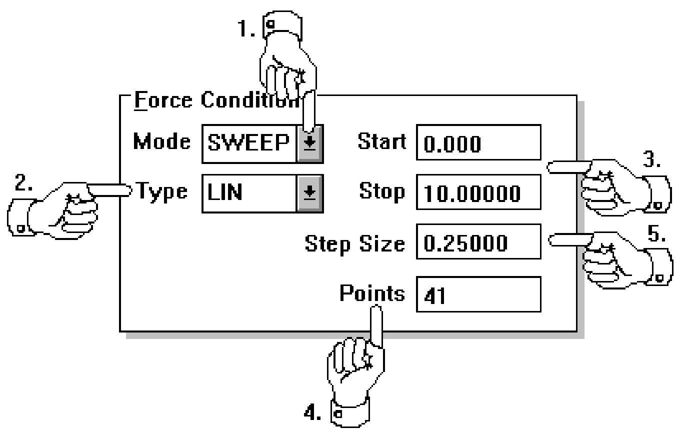


Figure 8: How to Force a Sweep Signal

How to Force a Sweep Signal:

1. Select the SWEEP designation from the available options listed in the MODE field.
2. The data point distribution of the sweep signal can be either linearly or logarithmically calculated between the boundary values. Select between a linear or logarithmic calculation by choosing the appropriate designation in the TYPE field.
3. Specify the sweep signal boundary values in the START and STOP.
4. Hitting the tab key from the STOP field will move the cursor to the POINTS field. Enter the number of data points that will comprise the sweep signal. If a logarithmic sweep type was selected, the value entered in the POINTS field will represent the total data point quantity of the sweep, not the data point quantity per decade.
5. If a linear sweep type was selected, the Force Conditions controls will include a STEP SIZE field. After a value is entered in the POINTS field and the cursor is moved to another location (or OK is selected), the STEP SIZE field will be calculated automatically. If desired, the calculated STEP SIZE can be updated by the user. If a new value is entered in the STEP SIZE field, the STOP field will be updated to accommodate the new STEP SIZE value. If any of the other fields are edited, the STEP SIZE field will be automatically updated to accommodate the change.

Step Mode

The step mode can only be used in test setups that also include the specification of a sweep signal. An instrument supply configured in a step mode will force a constant output while a second instrument supply outputs a sweep signal. At the completion of the sweep signal, the step value will be incremented in preparation for another sweep. This process continues until the stop value of the step signal is reached.

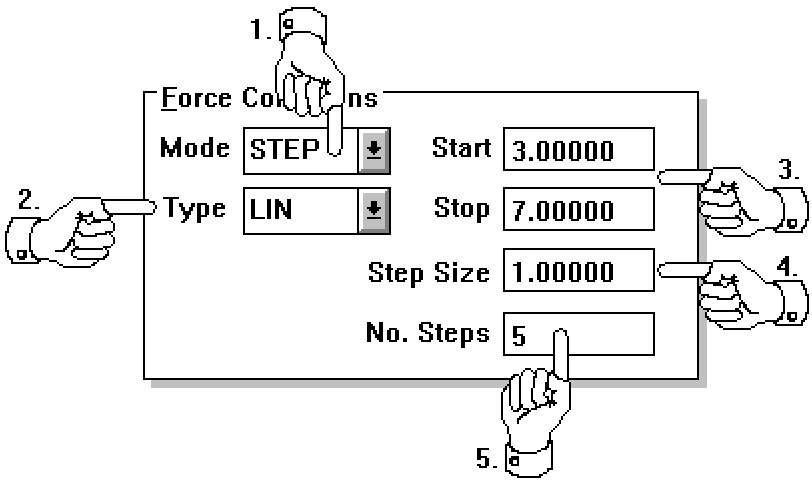


Figure 9: How to Force a Step Signal

How to Force a Step Signal:

1. Select the STEP designation from the available options listed in the MODE field.
2. The step increment can be either linearly or logarithmically calculated between the boundary values of the step signal. Select between either a linear or logarithmic step increment by choosing the appropriate designation in the TYPE field.
3. Specify the step signal boundary values in the START and STOP fields.
4. Hitting the tab key from the STOP field will move the cursor to the NO. STEPS field. Enter the step quantity.
5. If a linear sweep type was selected, the Force Conditions controls will include a STEP SIZE field. After a value is entered in the NO. STEPS field and the cursor is moved to another location (or OK is selected), the STEP SIZE field will be calculated automatically. If desired, the calculated STEP SIZE can be updated by the user. If a new value is entered in the STEP SIZE field, the STOP field will be updated to accommodate the new STEP SIZE value. If any of the remaining fields are edited, the STEP SIZE field will be automatically updated to accommodate the change. A measurement will take longer when a STEP SIZE value unavailable from the front panel is selected. ICS measures non-standard STEP SIZE curves by generating each curve as a separate single-step measurement. This methodology is transparent to the user except for the added time necessary to execute the measurement.

Constant Mode

An instrument supply configured in constant mode forces an output that remains at a constant magnitude throughout the duration of the test setup.

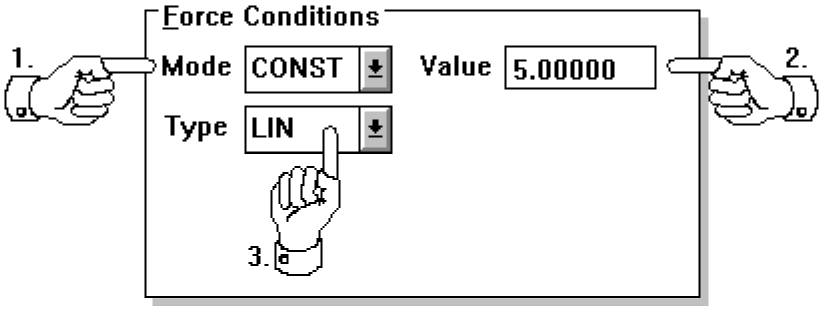


Figure 10: How to Force a Constant Signal

How to Force a Constant Signal:

1. Select the "CONST" designation from the available options listed in the MODE field.
2. Specify the signal magnitude in the VALUE field.
3. When configured in constant mode, the TYPE field is inactive.

Type

The TYPE field is used to apply either a linear or logarithmic characteristic to the forcing signal selected in the MODE field. If a sweep signal is selected in the MODE field, the characteristic selected in the TYPE field applies to the sweep signal data point distribution. If a step source is selected in the MODE field, the characteristic selected in the TYPE field applies to the step increment calculation.

Start

The START field is used to specify the starting value of the forcing signal. This field is present if a sweep or step signal is selected in the MODE field.

Stop

The STOP field is used to specify the final value of the forcing signal. This field is present if a sweep or step signal is selected in the MODE field. The STOP field is automatically updated when the STEP SIZE is specified by the user.

Step Size

The STEP SIZE field displays the step size in a linear sweep or step signal. This field is present if a sweep or step signal is selected in the MODE field, and a linear characteristic is selected in the TYPE field. The STEP SIZE is automatically calculated when the START, STOP, and either the NO. POINTS or NO. STEPS fields are specified. The STEP SIZE can also be specified by the user. When the STEP SIZE is specified by the user, the STOP field will be automatically updated to accommodate the change.

No. Points

The NO. POINTS field is used to specify the data point quantity in a linear or logarithmic sweep signal. This field is present if a sweep signal is selected in the MODE field, and a linear or logarithmic characteristic is selected in the TYPE field.

No. Steps

The NO. STEPS field is used to specify the step quantity in a linear or logarithmic step signal. This field is present if a step signal is selected in the MODE field, and a linear or logarithmic characteristic is selected in the TYPE field.

Value

The VALUE field is used to specify the magnitude of a constant forcing signal. This field is present if a constant signal is selected in the MODE field.

Specialized Base Supply Controls

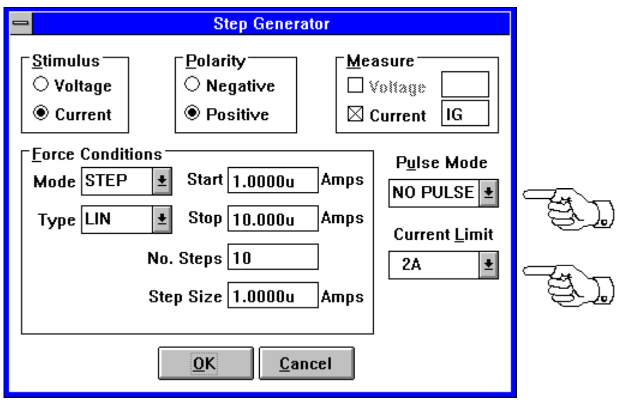


Figure 11: The Pulse Mode and Current Limit controls are specific to the Base Supply Dialog Box.

The Pulse Mode and Current Limit controls are specific to the Base Supply dialog box. The Pulse Mode and Current Limit options available from the Base Supply dialog box are identical to the options available from the instrument's front panel. Click on the scroll arrow to display a menu of available choices; click on the desired option.

Specialized Collector Supply Controls

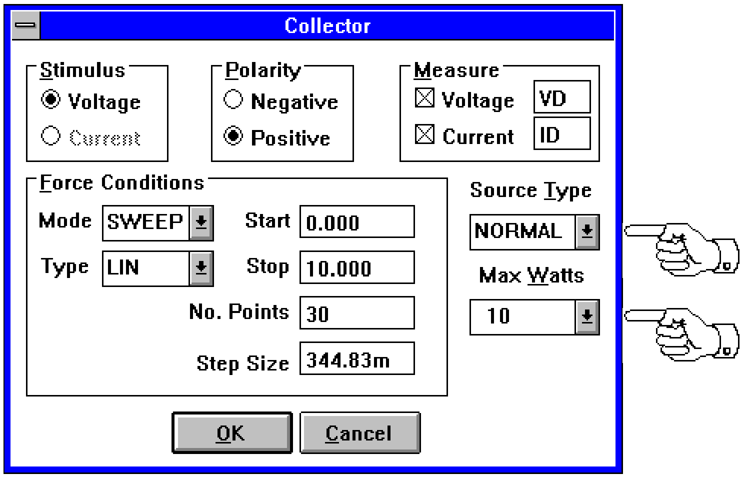
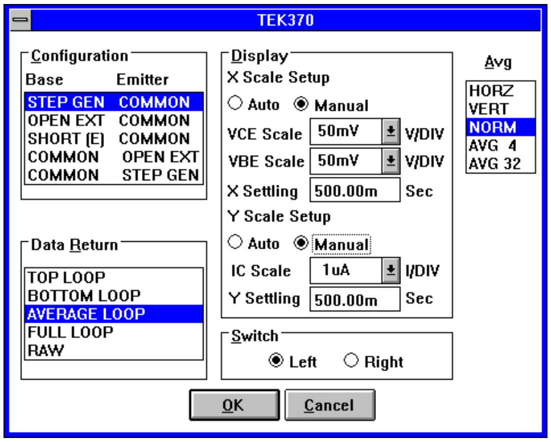


Figure 12: The Source Type and Max Watts controls are specific to the Collector Supply dialog box.

The Source Type and Max Watts controls are specific to the Collector Supply dialog box. The Source Type and Max Watts options available from the Collector Supply dialog box are identical to the options available from the instrument's front panel. The Source Type field provides the signal options available on the instrument's front panel Collector Supply Polarity switch. Click on the scroll arrow to display a menu of available choices; click on the desired option. When selecting a Source Type option, recall that polarity is specified with the Polarity Group controls included at the top of the dialog box.

Mainframe Controls

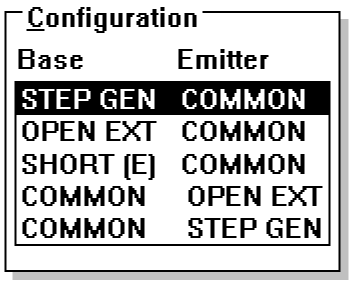


Clicking the Setup Editor OPTIONS button will open the Mainframe Controls dialog box. The Mainframe Controls dialog box includes the controls necessary to specify the instrument configuration, data return mode, instrument display scaling mode, standby switch position, as well as the data averaging mode.

How to Display the Mainframe Controls Dialog Box:

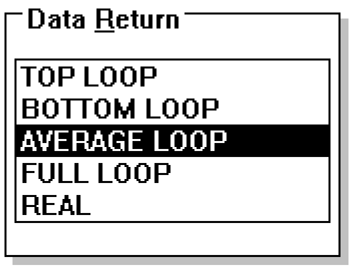
1. Run the Setup Editor by selecting the Setup Editor toolbar button. If creating a new test setup, be sure to specify a test setup name when prompted.
2. If at least one instrument supply has already been connected to a DUT pin, skip to Step #3. If no instrument supplies are connected, select the SOURCES button and assign an instrument supply to at least one DUT location. Close the Source Units dialog box.
3. From the Setup Editor, click the OPTIONS button.

Instrument Configuration



Select the desired instrument configuration by clicking on the appropriate option listed in the Configuration switch.

Data Return

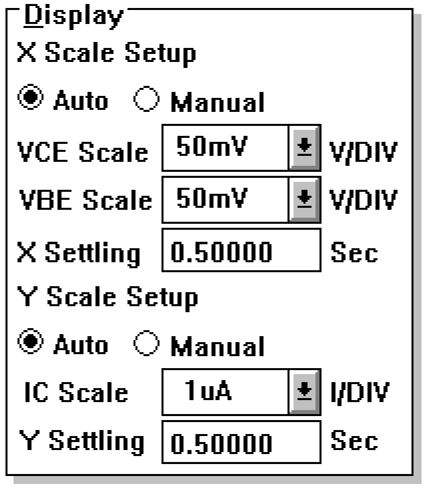


To eliminate the hysteresis characteristic in measured sweep curves, ICS allows you to specify which portion of the sweep curve to return as measured results. In addition to the real mode offered from the instruments' front panels, you can select from top loop, bottom loop, average loop, and full loop data return modes.

When the Tek370 executes a sweep measurement, the curve is actually created by sweeping from the START value to the STOP value, and then back from the STOP value to the START value. When temperature effects are present, the positive-going and negative-going sweep components induce a hysteresis characteristic to the measured sweep curves.

To eliminate this effect, the Tek370 Driver provides the capability to specify which portion of the sweep curve to return as measured results. The user can select from among TOP LOOP, BOTTOM LOOP, AVERAGE LOOP, FULL LOOP, or REAL LOOP data return modes. FULL LOOP will return both the top and bottom portions of a sweep curve while preserving each sweep as a unique data vector. REAL LOOP mode will return both the top and bottom portions of a sweep curve but will not recognize each sweep as a unique data vector. REAL LOOP mode is the data return mode implemented by the instrument's front panel display.

Display Scaling Options



ICS obtains data by digitizing the instrument's front panel display. While graphic results can be rescaled in a ICS plot window, the initial data resolution is determined by the horizontal and vertical deflection factors of the instrument. When both the horizontal and vertical autoscaling options are selected in the Display controls, ICS will optimize the resolution of the returned data by automatically searching through various horizontal and vertical deflection factors.

Each axis of the instrument's display can be independently configured in either an auto or manual scaling mode. When the manual scaling switch is selected, a deflection factor must be selected from the corresponding list box. To display a window of available options, click the list box down arrow. The deflection factors available in manual mode are the same deflection factors available from the front panel of the instrument. When manually scaling the instrument's horizontal axis, select the desired deflection factor from the list box corresponding to the source of the measured voltage, either the collector (Vce) or base (Vbe) supply.

Settling Time

The settling time specification in the Tek370 Driver is related to the autoscaling capability. When the instrument selects a deflection factor, the instrument will read the curve positions in order to evaluate the functionality of the selected deflection factor; however, it takes the instrument a finite time to refresh the display with the new condition. The settling time is the time the instrument will wait between selecting a deflection factor and reading the curve positions.

Occasionally, due to the complexity of the measurement, the refresh of the display will take longer than the default settling time, and the autoscaling feature will select a deflection factor that is too small resulting in a loss of data. The Settling field allows the user to vary the settling time specification to accommodate such measurements.

Standby Switch

Configure the Mainframe Control Standby switch so that it is in the same position as the instrument's front panel Standby switch.

Acquisition Controls

Acquisition Mode determines data acquisition mod for storage display. This setting does not affect NON STORE, VIEW or REF curves. Modes are;

HORIZ Envelope: when HORIZ Envelope is selected, the horizontal min/max envelopes of the waveform are displayed. The envelope display continuously accumulates until a setting changes.

VERT Envelope; WHEN VERT Envelope is selected, the vertical min/max envelopes of the waveform are displayed. The envelope display continuously accumulates until a setting changes.

NORM: Conventional digital storage function.

AVG: Selects 16 times averaging for accurate and precise measurement. This feature provides a filter effect to reduce display noise. The acquisition count is displayed in the upper-right graticule area.