Model 6482 Dual-Channel Picoammeter / Voltage Source

Reference Manual

6482-901-01 Rev. A / August 2012

KEITHLEY

A G R E A T E R M E A S U R E O F C O N F I D E N C E

Model 6482
Dual-Channel Picoammeter / Voltage Source
Reference Manual

Table of Contents

The following safety precautions should be observed before using this product and any associated instrumentation. Although some instruments and accessories would normally be used with nonhazardous voltages, there are situations where hazardous conditions may be present.

This product is intended for use by qualified personnel who recognize shock hazards and are familiar with the safety precautions required to avoid possible injury. Read and follow all installation, operation, and maintenance information carefully before using the product. Refer to the user documentation for complete product specifications.

If the product is used in a manner not specified, the protection provided by the product warranty may be impaired.

The types of product users are:

Responsible body is the individual or group responsible for the use and maintenance of equipment, for ensuring that the equipment is operated within its specifications and operating limits, and for ensuring that operators are adequately trained.

Operators use the product for its intended function. They must be trained in electrical safety procedures and proper use of the instrument. They must be protected from electric shock and contact with hazardous live circuits.

Maintenance personnel perform routine procedures on the product to keep it operating properly, for example, setting the line voltage or replacing consumable materials. Maintenance procedures are described in the user documentation. The procedures explicitly state if the operator may perform them. Otherwise, they should be performed only by service personnel.

Service personnel are trained to work on live circuits, perform safe installations, and repair products. Only properly trained service personnel may perform installation and service procedures.

Keithley Instruments products are designed for use with electrical signals that are rated Measurement Category I and Measurement Category II, as described in the International Electrotechnical Commission (IEC) Standard IEC 60664. Most measurement, control, and data I/O signals are Measurement Category I and must not be directly connected to mains voltage or to voltage sources with high transient over-voltages. Measurement Category II connections require protection for high transient over-voltages often associated with local AC mains connections. Assume all measurement, control, and data I/O connections are for connection to Category I sources unless otherwise marked or described in the user documentation.

Exercise extreme caution when a shock hazard is present. Lethal voltage may be present on cable connector jacks or test fixtures. The American National Standards Institute (ANSI) states that a shock hazard exists when voltage levels greater than 30 V RMS, 42.4 V peak, or 60 V DC are present. A good safety practice is to expect that hazardous voltage is present in any unknown circuit before measuring.

Operators of this product must be protected from electric shock at all times. The responsible body must ensure that operators are prevented access and/or insulated from every connection point. In some cases, connections must be exposed to potential human contact. Product operators in these circumstances must be trained to protect themselves from the risk of electric shock. If the circuit is capable of operating at or above 1000 V, no conductive part of the circuit may be exposed.

Do not connect switching cards directly to unlimited power circuits. They are intended to be used with impedance-limited sources. NEVER connect switching cards directly to AC mains. When connecting sources to switching cards, install protective devices to limit fault current and voltage to the card.

Before operating an instrument, ensure that the line cord is connected to a properly-grounded power receptacle. Inspect the connecting cables, test leads, and jumpers for possible wear, cracks, or breaks before each use.

When installing equipment where access to the main power cord is restricted, such as rack mounting, a separate main input power disconnect device must be provided in close proximity to the equipment and within easy reach of the operator.

For maximum safety, do not touch the product, test cables, or any other instruments while power is applied to the circuit under test. ALWAYS remove power from the entire test system and discharge any capacitors before: Connecting or disconnecting cables or jumpers, installing or removing switching cards, or making internal changes, such as installing or removing jumpers.

Do not touch any object that could provide a current path to the common side of the circuit under test or power line (earth) ground. Always make measurements with dry hands while standing on a dry, insulated surface capable of withstanding the voltage being measured.

The instrument and accessories must be used in accordance with its specifications and operating instructions, or the safety of the equipment may be impaired.

Do not exceed the maximum signal levels of the instruments and accessories, as defined in the specifications and operating information, and as shown on the instrument or test fixture panels, or switching card.

When fuses are used in a product, replace with the same type and rating for continued protection against fire hazard.

Chassis connections must only be used as shield connections for measuring circuits, NOT as safety earth ground connections.

If you are using a test fixture, keep the lid closed while power is applied to the device under test. Safe operation requires the use of a lid interlock.

If a ⏚ screw is present, connect it to safety earth ground using the wire recommended in the user documentation.

The ⚠ symbol on an instrument means caution, risk of danger. The user should refer to the operating instructions located in the user documentation in all cases where the symbol is marked on the instrument.

The ⚡ symbol on an instrument means caution, risk of electric shock. Use standard safety precautions to avoid personal contact with these voltages.

The ♨ symbol on an instrument shows that the surface may be hot. Avoid personal contact to prevent burns.

The ⏚ symbol indicates a connection terminal to the equipment frame.

If this (Hg) symbol is on a product, it indicates that mercury is present in the display lamp. Please note that the lamp must be properly disposed of according to federal, state, and local laws.

The **WARNING** heading in the user documentation explains dangers that might result in personal injury or death. Always read the associated information very carefully before performing the indicated procedure.

The **CAUTION** heading in the user documentation explains hazards that could damage the instrument. Such damage may invalidate the warranty.

Instrumentation and accessories shall not be connected to humans.

Before performing any maintenance, disconnect the line cord and all test cables.

To maintain protection from electric shock and fire, replacement components in mains circuits - including the power transformer, test leads, and input jacks - must be purchased from Keithley Instruments. Standard fuses with applicable national safety approvals may be used if the rating and type are the same. Other components that are not safety-related may be purchased from other suppliers as long as they are equivalent to the original component (note that selected parts should be purchased only through Keithley Instruments to maintain accuracy and functionality of the product). If you are unsure about the applicability of a replacement component, call a Keithley Instruments office for information.

To clean an instrument, use a damp cloth or mild, water-based cleaner. Clean the exterior of the instrument only. Do not apply cleaner directly to the instrument or allow liquids to enter or spill on the instrument. Products that consist of a circuit board with no case or chassis (e.g., a data acquisition board for installation into a computer) should never require cleaning if handled according to instructions. If the board becomes contaminated and operation is affected, the board should be returned to the factory for proper cleaning/servicing.

Getting Started

In this section:

Introduction

This section contains basic information about the Keithley Instruments Model 6482 Dual-Channel Picoammeter / Voltage Source. The information is organized as follows:

- General information
- Product overview
- Specifications
- Front and rear panel familiarization
- Power-up
- Display
- Default settings
- Menus

General information

This section covers general information that includes warranty information, contact information, safety symbols and terms, inspection, and available options and accessories.

Extended warranty

Additional years of warranty coverage are available on many products. These valuable contracts protect you from unbudgeted service expenses and provide additional years of protection at a fraction of the price of a repair. Extended warranties are available on new and existing products. Contact your local Keithley Instruments representative for details.

Inspection for damage

The Model 6482 was carefully inspected electrically and mechanically before shipment. After unpacking all items from the shipping carton, check for any obvious signs of physical damage that may have occurred during transit. There may be a protective film over the display lens, which can be removed. Report any damage to the shipping agent immediately. Save the original packing carton for possible future shipment.

Shipment contents

The following items are included with every Model 6482 Dual-Channel Picoammeter order:

- Model 6482 with line cord
- Model 7078-TRX-BNC adapters (quantity 2)
- Accessories as ordered
- Certificate of calibration
- One CD-ROM (including this reference manual)

Instruction manual

A CD-ROM containing this reference manual and any required software and drivers is included with each Model 6482 order.

Repacking for shipment

If you need to return the Model 6482 for repair, carefully pack the instrument in its original packing carton or the equivalent, and follow these instructions:

- Call the Repair Department at 1-888-KEITHLEY (1-888-534-8453) for a Return Material Authorization (RMA) number.
- Advise as to the warranty status of the Model 6482.
- Write ATTENTION REPAIR DEPARTMENT and the RMA number o n the shipping label.

Options and accessories

The following options and accessories are available from Keithley for use with the Model 6482.

Triaxial cables and adapters

Model 7078-TRX-1: This low-noise 12-inch triaxial cable is terminated at both ends with 3-slot male triaxial connectors.

Model CS-751 Barrel Adapter: This is a barrel adapter that allows you to connect two triaxial cables together. Both ends of the adapter are terminated with 3-lug female triaxial connectors.

Model CS-1053 Barrel Adapter: This barrel adapter is terminated at both ends with 3-slot male triaxial connectors.

Model 237-BNC-TRX Adapter: This is a male BNC to 3-lug female triaxial adapter (guard disconnected). It is used to terminate a triaxial cable with a BNC plug.

Model 237-TRX-T Adapter: This is a 3-slot male to dual 3-lug female triaxial tee adapter for use with triaxial cables.

Model 7078-TRX-BNC Adapter: This is a 3-slot male triaxial to female BNC adapter. This adapter lets you connect a BNC cable to the triaxial input of the Model 6482.

Model 237-TRX-TBC Connector: This is a 3-lug female triaxial bulkhead connector with cap for assembly of custom panels and interface connections.

Interface cables

Models 7007-1 and 7007-2 shielded GPIB cables: Connect the Model 6482 to the GPIB bus using shielded cables and connectors to reduce electromagnetic interference (EMI). The Model 7007-1 is 1 m (3.3 ft) long; the Model 7007-2 is 2 m (6.6 ft) long.

Model 7009-5 shielded RS-232 cable: Connect the Model 6482 to computer serial port using shielded cables and connectors to reduce EMI. The Model 7009-5 is 1.5 m (5 ft) long.

Models 8501-1 and 8501-2 trigger link cables: Connect the Model 6482 to other instruments with trigger link connectors (for example, Model 7001 Switch System). The Model 8501-1 is 1 m (3.3 ft) long; the Model 8501-2 is 2 m (6.6 ft) long.

Model 8502 trigger link adapter: Connects any of the six trigger link lines of the Model 6482 to instruments that use the standard BNC trigger connectors.

Model 8503 DIN to BNC trigger cable: Connects the trigger link lines one (voltmeter complete) and two (external trigger) of the Model 6482 to instruments that use BNC trigger connectors. The Model 8503 is 1 m (3.3 ft) long.

Rack mount kits

Model 4288-1 single fixed rack mount kit: Mounts a single Model 6482 (or any of the following: Models 2000, 2001, 2002, 2010, 2015, 2016, 2400, 2410, 2420, 2430, 2500, 2510, 6430, 6482, 6485, 6517A, 7001) in a standard 19-inch rack.

Model 4288-2 side-by-side rack mount kit: Mounts two instruments (Models 428-PROG, 2000, 2001, 2002, 2010, 2015, 2016, 2400, 2410, 2420, 2430, 2500, 2510, 6430, 6482, 6485, 6517A, 7001) next to each other in a standard 19-inch rack.

Model 4288-5/Model 4288-9 dual fixed rack mounting kit (shelf-type mount): Mounts a Model 6482 and another ½-inch high instrument (Models 428-PROG, 2000, 2010, 2400, 2410, 2420, 2430, or 7001) next to each other in a standard 19-inch rack. The Model 4288-5 includes standard-length rear support brackets while the Model 4288-9 includes extended-length rear support brackets.

Model 4299-7 fixed rack mounting kit (shelf-type mount): Mounts a single Model 6482 and a front-panel plate, or two instruments (Series 2000, 2100, 2200, 2300, 2400, 2500; Models 428-PROG, 2182A, 2700, 2701, 3390, 6220, 6221, 6430, 6482, 6485, 6487, 6514, 6517B, 7001) side-by-side in a standard 19-inch rack. This kit also supports the following Tektronix bench instruments: Series PWS2000, PWS4000 (power supplies); DMM4020, DMM4040, DMM4050 (digital multimeters).

Carrying case

Model 1050 padded carrying case: A carrying case for a Model 6482. Includes handles and a shoulder strap.

Product overview

This section summarizes the features of the Model 6482 Dual-Channel Picoammeter.

The Model 6482 combines two stable DC bias voltage sources with two low-noise picoammeters for dual-channel measurements. The instrument has 0.1 percent basic measurement accuracy with up to 6½-digit resolution. At 5½-digit resolution, the Model 6482 can transfer up to 475 readings per second over the GPIB connection, and up to 900 readings per second with 4½-digit resolution.

Each Model 6482 channel has the following basic measurement and voltage bias capabilities:

- Measure current from 1 fA to 20 mA.
- Source bias voltage from 500 µV to 30 V.

Additional capabilities of the Model 6482 include:

- Sweep capabilities: Linear and logarithmic staircase, and custom sweeps of up to 2500 points.
- Two hardware limit tests and four software limit tests with a built-in comparator for pass/fail testing.
- Programming language and remote interfaces: The Model 6482 uses the SCPI programming language and two remote interface ports (IEEE-488/GPIB and RS-232C).
- Trigger-link interface to Keithley Series 7000 switching hardware.
- Math functions: V/I, I/V, electrical power, ratio, delta, and offset functions.
- Reading and setup storage: Up to 3000-point buffer on each channel that can contain up to 5½-digit readings, and seven setups (five user defaults, factory default, *RST default) that can be stored and recalled.
- Calibration: The instrument can be calibrated from the front-panel or a remote interface.
- An analog output for each channel. Each analog output provides a -10 V to +10 V output voltage analogous to the input current.

Specifications

Front and rear panel familiarization

This section summarizes the controls and connectors on the instrument.

Front-panel summary

The front panel of the Model 6482 is shown in Figure 1-1.

Figure 1-1
Front panel

Model 6482

Measurement function keys:

MSR1	Measure channel 1.
MSR2	Measure channel 2.
DELTA	Difference function.
RATIO	Ratio function.

Voltage bias source function keys:

SRC1	Channel 1 voltage bias source.
SRC2	Channel 2 voltage bias source.

EDIT keys:

▲	Increase value.
▼	Decrease value.
◀	Move cursor left.
▶	Move cursor right.

Range keys:

AUTO	Enable or disable autorange.
▲	Increase manual range.
▼	Decrease manual range.

Operation keys:

DISPLAY CHANNEL SELECT	Select channel 1 or channel 2 measurement for display.
DISPLAY TOGGLE	Toggle between channel 1 and 2 display (add MSR2 to display for dual display mode).
LOCAL	Cancel remote operation.
REL	Enable or disable relative reading (MSR1 or MSR2 function only).
FILTER	Display digital filter status for present function and toggle filter on and off.
LIMIT	Perform configured limit tests.
TRIG	Trigger a measurement from the front panel.
SWEEP	Start a configured sweep.
DIGITS	Change the number of digits of display resolution.
SPEED	Change measurement speed by selecting accuracy or specifying NPLC.
STORE	Set buffer size and enable reading storage.
RECALL	Display stored readings and timestamp.
CONFIG	Press CONFIG and then appropriate key to configure function or operation.
MENU	Access and configure Main Menu selections. When entering numeric data, use to clear reading to minimum absolute value.
EXIT	Cancels selection. Use to return to the previous menu.
ENTER	Accepts selection.

Indicators:

EDIT	Instrument is in edit mode.
ERR	Questionable reading, invalid calibration step.
REM	Instrument is in GPIB remote mode.
TALK	Instrument is addressed to talk over GPIB.
LSTN	Instrument is addressed to listen over GPIB.
SRQ	Service request over GPIB.
REL	Relative measure reading displayed.
FILT	Digital filter enabled.
AUTO	Autoranging enabled.
ARM	Source-measure operations being performed.
TRIG	External trigger source selected.
*	Reading being stored.

Power control:

POWER	Turns the instrument power on or off.
OUTPUT ON/OFF	Turns the sources on or off.

Handle:

Pull out and rotate.

Rear panel summary

NOTE   *For additional information about connectors, see Connector terminals.*

The rear panel of the Model 6482 is shown in Figure 1-2.

Figure 1-2
Rear panel

Model 6482
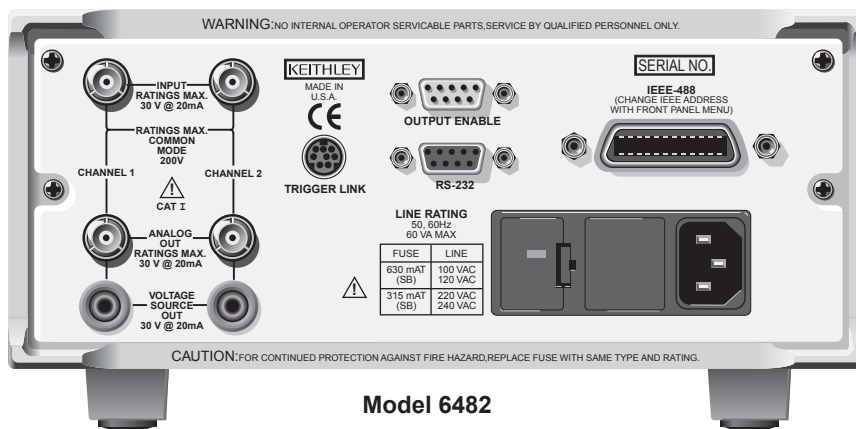
WARNING   ***Maximum common-mode voltage (voltage between INPUT common and chassis ground) is 200 V. Exceeding this value may result in a shock hazard.***

CAUTION   **Maximum INPUT and OUTPUT connector rating is 30 V at 20 mA. Exceeding these values may result in instrument damage.**

INPUT connectors:

INPUT CHANNEL 1	Use to measure on channel 1.
INPUT CHANNEL 2	Use to measure on channel 2.
Triaxial connector terminals:	Center conductor – Input HI.
	Inner shield – Floating input/output common.
	Outer shield – Chassis ground.

ANALOG OUT connectors:

ANALOG OUT	Triaxial connectors for CHANNEL 1 and CHANNEL 2 analog outputs.
Triaxial connector terminals:	Center conductor – Analog output HI.
	Inner shield – Analog output LO (analog common)
	Outer shield – Chassis ground.

VOLTAGE SOURCE connectors:

OUTPUT CHANNEL 1	Use to output voltage bias on channel 1.
OUTPUT CHANNEL 2	Use to output voltage bias on channel 2.

Power module:

Contains the AC line receptacle and the power line fuse.

Trigger link connector:

TRIGGER LINK	8-pin micro-DIN connector for sending and receiving trigger pulses. Use a trigger link cable or adapter, such as Models 8501-1, 8501-2, 8502, 8504.

RS-232 connector:

RS-232   Connector for RS-232 remote operation. Use a straight through (not
                                    null modem) DB-9 cable such as a Keithley Model 7009-5.

OUTPUT ENABLE connector:

OUTPUT ENABLE   Connector for output enable operation. Male DB-9 connector for output
                                    enable line and ground.

GPIB connector:

IEEE-488 interface   Connector for GPIB remote operation. Use a shielded cable
                                    (Model 7007-1 or 7007-2).

Power-up

This section covers line power connection, line voltage settings, fuse replacement, and the power-up sequence.

Line voltage selection

The Model 6482 operates from a line voltage in the range of 100 V, 120 V, 220 V, 240 V (±10 percent) at a frequency of 50 Hz or 60 Hz. Before plugging in the instrument, make sure the line voltage setting visible through the small window in the power module (Figure 1-2) is correct for the line voltage in your area.

> **CAUTION**   **Operating the instrument on an incorrect line voltage may cause damage, possibly voiding the warranty.**

If the voltage setting is not correct, change it as described in Section 18.

Line power connection

Perform the following steps to connect the Model 6482 to line power and turn it on:

1. Before plugging in the power cord, make sure the front-panel power switch is in the off (0) position.
2. Connect the female end of the supplied power cord to the AC receptacle on the rear panel.

> **WARNING**   **The power cord supplied with the Model 6482 contains a separate ground for use with grounded outlets. When proper connections are made, the instrument chassis is connected to power line ground through the ground wire in the power cord. Failure to use a grounded outlet may result in personal injury or death due to electric shock.**

3. Turn on the instrument by pressing the front-panel power switch to the on (1) position.

Power-up sequence

When the Model 6482 is turned on, it performs self-tests on its EPROM and RAM and momentarily lights all segments and indicators. If a failure is detected, the instrument momentarily displays an error message, and the ERR indicator turns on. Error messages are listed in Appendix A.

> **NOTE**   *If a problem develops while the instrument is under warranty, return it to Keithley Instruments for repair.*

If the instrument passes the self-tests, the firmware revision levels are displayed. For example:

```
Rev A01    A02
```

Where: A01 is the main board ROM revision.
          A02 is the display board ROM revision.

Also displayed is the line frequency (50 Hz or 60 Hz). If the wrong frequency is displayed, it can be set manually as described below. The communication interface status is briefly displayed. If the IEEE-488 bus is the presently selected interface, the identification message includes the primary address. For example, if the primary address is 25 (factory default), the `SCPI Addr=25` message is displayed. If the RS-232 interface is selected, the `RS-232 mode` message is displayed.

After the power-up sequence, the instrument goes to its normal display state with the output off (the blue ON/OFF OUTPUT indicator only lights when the output is on).

System identification

To obtain the serial number and revision information, press the **MENU** key, and then select **GENERAL** > **SERIAL #** or send the *IDN? query from a remote interface.

Line frequency setting

If the line frequency setting (50 Hz or 60 Hz) does not match the power line frequency in your area, noisy measurement readings will result, and accuracy may be affected. You can manually set the line frequency from the front panel or through a remote interface as described below. Make sure to also check the instruments power line voltage setting (see Line voltage selection).

CAUTION   **Operating the instrument on an incorrect line voltage may cause damage, possibly voiding the warranty.**

Front-panel line frequency

To set the line frequency from the front panel:

1. Press the **MENU** key to display MAIN MENU.
2. Press the EDIT ◀ or ▶ keys to select **AD-CTRL**, and then press the **ENTER** key to display the A/D CONTROLS menu.
3. Select **LINE-FREQUENCY**, and then press the **ENTER** key to display the LINE FREQUENCY menu.
4. Place the cursor on **50 Hz** or **60 Hz**, and then press the **ENTER** key.
5. Press the **EXIT** key as needed to return to normal display.

Remote command line frequency

Use the following command to set the line frequency:

```
:SYSTem:LFRequency <Freq>
```

See Section 16 for additional information on using this command.

Programming example

50 Hz operation:

```
:SYST:LFR 50
```

Fuse replacement

A rear-panel fuse protects the power line input of the Model 6482. If the line fuse needs to be replaced, replace it as described in Section 18.

Display

This section provides information about the Model 6482 display.

Display format

The Model 6482 display is used primarily to display measured readings and voltage bias source values. The measurement function readings are displayed on the top line and the voltage bias values are displayed on the bottom line. You can select channel 1, channel 2, or dual-channel display with the CHANNEL SELECT and DISPLAY TOGGLE keys; see "DISPLAY CHANNEL SELECT key" and "DISPLAY TOGGLE key" below. When in dual-channel display mode, channel 1 measurement function readings (MSR1) are displayed on the left and channel 2 measurement function values (MSR2) appear on the right.

Measurement reading information can be displayed using either engineering units or scientific notation in either fixed- or floating-point format. To select the display format, press the **MENU** key, and then select **GENERAL** > **NUMBERS** (for more information, see "Menus" later in this section).

Engineering units example: 1.234567 µA

Scientific notation example: 1.234567e -6A

Indicators, which are located along the top of the reading/message display, indicate various states of operation, as covered previously in "Front-panel summary."

DISPLAY CHANNEL SELECT key

The CHANNEL SELECT key toggles between the channel 1 and channel 2 measurements (on the display's left side) when the single-channel display mode is selected.

DISPLAY TOGGLE key

The DISPLAY TOGGLE key turns the dual-channel display mode on or off. When the dual-channel mode is active, the channel 1 measurement function reading (MSR1) appears on the left (I, V/I, I/V, I $\times$ V, or MX + B) and the channel 2 measurement function (MSR2) appears on the right of the top line. The DISPLAY TOGGLE key is also used to select the channel and display statistical data about readings stored in the data store buffer. These functions are performed from the data store RECALL mode. See Section 6 for data store information.

Status and error messages

Status and error messages are displayed momentarily. During Model 6482 operation and programming, you will encounter a number of front-panel messages. Typical messages are either status messages or error messages and are displayed momentarily (see Appendix A for information).

Disabling the front-panel display

Front-panel display circuitry can be disabled to allow the instrument to operate at a higher speed (see Front-panel control). While disabled, the display is frozen with the following message:

```
FRONT PANEL DISABLED
```

```
Press LOCAL to resume.
```

As reported by the message, all front-panel controls (except LOCAL, TRIG, and OUTPUT ON/OFF) are disabled.

Front-panel control

The front-panel display circuitry is controlled from the DISABLE DISPLAY configuration menu, which is accessed by pressing the **CONFIG** key and then the **CHANNEL SELECT** or **DISPLAY TOGGLE** key. To select an option (NOW, NEVER, SWEEP, or STORE), use the ◀ and ▶ keys to place the cursor on the option and then press the **ENTER** key.

The options for DISABLE DISPLAY are:

NOW: Select this option to disable the display now.

NEVER: Select this option if you do not want the display to be disabled.

SWEEP: Select this option if you want the display to be disabled while performing a sweep. The display will disable as soon as sweep is started. The display will automatically re-enable after the sweep is completed.

STORE: Select this option if you want the display to be disabled when storing source-measure readings in the buffer. The display is disabled as soon as the buffer is enabled. The display is automatically re-enabled after the storage process is completed. Note that with this option, the display will disable while performing a sweep. Sweep readings are automatically stored in the buffer.

Remote display programming

The display can be controlled by various SCPI :DISPlay subsystem commands. Table 1-1 summarizes basic commands. See Section 16, "DISPlay subsystem," for more information on using these commands.

Table 1-1
Basic display commands

Command	Description

`:DISPlay:ENABle <state>`
`:DISPlay:DIGits <n>`
`:DISPlay:MODE <name>`
	Enable or disable display (state = ON or OFF).
	Set display resolution (n = 3 to 6).
	Select display mode (name = CALC3,
	    CALC4, CALC5, CALC6, or DUAL).

Front-panel tests

To perform various front-panel tests, press the **MENU** key and then select **TEST** > **DISPLAY TESTS**. Test selections include:

- **KEYS**: Front-panel keys are tested. Pressing a key displays a message that identifies that key. Press the **EXIT** key twice to cancel this test.
- **DISPLAY PATTERNS**: Use this selection to turn on all display pixels and indicators. Subsequent key presses cycle through tests that turn off indicators and corner pixels of each digit, turn on the rows of the top-left display digit, and turn on all indicators and pixels of each digit sequentially. Press the **EXIT** key to cancel this test.
- **CHAR SET**: This test displays special characters. Press the **EXIT** key to cancel the test.

See "Menus", for more menu information.

Default settings

This section covers factory default setups and saving and recalling user setups. By using appropriate menu selections, as described below you can save and restore various instrument setups, define the power-on configuration, or restore factory defaults.

Saving and restoring user setups

You can save and restore up to five of your own user setups using the following procedures.

Saving setups

1. Select the various instrument operating modes you wish to save.
2. Press the **MENU** key, select **SAVESETUP**, and then press the **ENTER** key.
3. From the SAVESETUP MENU, select **SAVE**, and then press the **ENTER** key.
4. Using the EDIT or RANGE ▲ or ▼ keys, select the setup position (0 to 4) to save, and then press the **ENTER** key to complete the process.

Restoring setups

1. Press the **MENU** key, select **SAVESETUP**, and then press the **ENTER** key.
2. From the SAVESETUP MENU, select **RESTORE**, and then press the **ENTER** key.
3. Using the EDIT or RANGE ▲ or ▼ keys, select the setup position (0-4) to restore, and then press the **ENTER** key to complete the process.

Power-on configuration

You can also define which of the stored setups (factory default or user) the instrument uses as the power-on configuration, as follows:

1. Press the **MENU** key, select **SAVESETUP**, and then press the **ENTER** key.
2. From the SAVESETUP MENU, select **POWERON**, and then press the **ENTER** key.
3. From the SET POWER-ON DEFAULT menu, choose the power-on configuration: **BENCH** or **GPIB** (see Table 1-2 for specifics), or **USER-SETUP-NUMBER**.
4. If you chose to use a user setup as the power-on configuration, select the user setup number, and then press the **ENTER** key.

Factory default settings

As summarized in Table 1-2, there are two sets of factory defaults: BENCH (front panel) and GPIB (remote). You can restore either of these default conditions as follows:

1. Press the **MENU** key, select **SAVESETUP**, and then press the **ENTER** key.
2. From the SAVE SETUP MENU, select **RESET**, and then press the **ENTER** key.

3. Select **BENCH** or **GPIB** defaults (see Table 1-2 for specifics), and then press the **ENTER** key to complete the process.

Table 1-2
Factory default settings

Setting*	BENCH default	GPIB default
A/D Controls:		
Auto-zero	Enable	On
Line frequency	No effect	No effect
Data store	No effect	No effect
Display digits	5.5	5.5
Display mode	Dual-channel	Dual-channel
Filter:	Off	Off
Average mode, state*	Repeat, disabled	Repeat, disabled
Average filter count*	10	10
Advanced state, tolerance. *	Off, 5%	Off, 5%
Median filter rank, state*	0, off	0, off
GPIB address	No effect	No effect
Ground connect	Disabled	Disabled
Limit tests:		
H/W** limits (1 and 2):		
Control	Disabled	Disabled
Fail-mode	In compliance	In compliance
S/W** limits (3 to 6):		
Control	Disabled	Disabled
Feed	Channel 1 measure	Channel 1 measure
Math function and units:	MX +B, MXB	MX + B, MXB
M factor (gain)	1	1
B factor (slope)	0	0
State	Disabled	Disabled
Numbers	No effect	No effect
Output:	Off	Off
Output enable	Disabled	Disabled

* Settings are for both channel 1 and channel 2.
** H/W: hardware; SW: software

Table 1-2 (cont.)
Factory default settings

Setting*	BENCH default	GPIB default
Power-on default	No effect	No effect
Ranging (measure):		
Autorange	Enabled	Enabled
Upper limit	20 mA	20 mA
Lower limit	2 nA	2 nA
Rel:	Off	Off
Value	0.0	0.0
RS-232	No effect	No effect
Source delay:	1 ms	1 ms
Autodelay	Enabled	Enabled
Triggering:		
Arm layer:		
Event	Immediate	Immediate
Count	1	1
Timer	0.1	0.1
Input line	1	1
Output trigger	Line #2, off	Line #2, off
Trigger layer:		
Event	Immediate	Immediate
Count	1	1
Timer	0.1	0.1
Input line	1	1
Output triggers	Line #2, all off	Line #2, all off
Delay	0.0 sec	0.0 sec
Voltage bias sources:		
Amplitude	0 V	0 V
Range	10 V	10 V
Mode	Fixed	Fixed
Speed	1 PLC	1 PLC
Sweep:	None	None
Start	0 V	0 V
Stop	0 V	0 V
Step	0 V	0 V
Direction	Up	Up
Sweep count	1	1
Sweep points	3000	3000
Source ranging	Best fixed	Best fixed
Triggered source:		
Control	Disabled	Disabled
Amplitude	0	0

* Settings are for both channel 1 and channel 2.
** H/W: hardware; SW: software

Remote setups

You can also save and recall setups from a remote interface using the following SCPI commands:

- Save and recall user setups using *SAV and *RCL (see Section 14).
- Restore GPIB defaults using *RST (see Section 14).
- Restore bench defaults using :SYSTem:PRESet (see Section 16).
- Save the power-on configuration using :SYSTem:POSetup (see Section 16).

Menus

The following paragraphs discuss the main menu, configuration menus, and rules to navigate menus.

Main menu

Use the MENU key to access the main menu to select, configure, and/or perform various instrument operations. These include default setup conditions, communications (GPIB or RS-232), calibration, front-panel tests, auto zero, timestamp, and numeric display format.

The main menu structure is summarized in Table 1-3. Use the "Rules to navigate menus" to check and change menu options. Figure 1-3 shows the main menu tree.

Table 1-3
Main menu

Menu item[1]	Description	Parameters
SAVESETUP	Configure setup conditions.	
SAVE	Save present Model 6482 setup to memory location.	0 to 4
RESTORE	Return the Model 6482 to setup saved in memory.	0 to 4
POWERON	Select the power-on default setup.	
BENCH	Powers-on to BENCH defaults.	See Table 1-2.
GPIB	Powers-on to GPIB defaults.	See Table 1-2.
USER-SETUP-NUMBER	Powers-on to user setup.	0 to 4
RESET	Returns instrument to BENCH or GPIB defaults.	See Table 1-2.
COMMUNICATION[2]	Select and configure the remote interface.	
GPIB	Select GPIB (IEEE-488 bus), set primary address, GPIB protocol (see Appendix E).	0 to 30 (Default: 25)
RS-232	Select the RS-232 interface, set parameters.	
BAUD	Select baud rate.	57600, 38400, 19200, 9600, 4800, 2400, 1200, 600, 300
BITS	Select number of data bits.	7 or 8
PARITY	Select parity.	NONE, ODD, EVEN
TERMINATOR	Select terminator.	CR, CR+LF, LF, or LF+CR
FLOW-CTRL	Select flow control.	NONE or XON/XOFF

Notes:
[1] Top-level menu choices are indicated in bold. Indentation identifies each lower submenu level.
[2] When the remote operation interface selection (GPIB or RS-232) is changed, the Model 6482 performs a power-on reset. To check or change options of the selected interface, you must then re-enter the menu structure.
[3] A password is required to unlock calibration. See the "Calibration" section.
[4] Press the **EXIT** key to cancel the test.
[5] Disabling auto zero will reduce measurement accuracy.

Table 1-3 (cont.)
Main menu

Menu item[1]	Description	Parameters
CAL[3]	Calibrate Model 6482. See "Calibration" for menu structure.	
TEST	Perform tests on Model 6482.	
DISPLAY-TESTS[4]	Test front-panel keys and display digits.	
KEYS	Test front-panel keys.	
DISPLAY-PATTERNS	Test display pixels and indicators.	
CHAR SET	Test special display characters.	
A/D CONTROLS	Control autozero, line frequency, and NPLC caching.	
AUTO-ZERO[5]	Control autozero.	
DISABLE	Disable autozero.	
ENABLE	Enable autozero.	
LINE-FREQUENCY	Set the line frequency.	50 Hz or 60 Hz
GENERAL	Select general operations.	
SERIAL#	Display serial number, firmware revision, SCPI version.	
TIMESTAMP	Reset timestamp.	YES or NO
NUMBERS	Select engineering units or scientific notation display format.	ENGR, SCIENTIFIC

Notes:
[1] Top-level menu choices are indicated in bold. Indentation identifies each lower submenu level.
[2] When the remote operation interface selection (GPIB or RS-232) is changed, the Model 6482 performs a power-on reset. To check or change options of the selected interface, you must then re-enter the menu structure.
[3] A password is required to unlock calibration. See the "Calibration" section.
[4] Press the **EXIT** key to cancel the test.
[5] Disabling auto zero will reduce measurement accuracy.

Rules to navigate menus

Many functions and operations are configured from the front-panel menus. Use the following rules to navigate through these configuration menus:

- A menu item is selected by placing the cursor on it and pressing the **ENTER** key. Cursor position is denoted by the blinking menu item or option. The EDIT ◀ or ▶ keys control cursor position.
- A displayed arrow on the bottom line indicates there are one or more additional items (or messages) to select from. Use the appropriate cursor key to display them.
- A measurement or voltage bias source range is changed by selecting the channel and function with one of the MSR1, MSR2, SRC1, or SRC2 keys, and then using the RANGE ▲ or ▼ keys. Note that when the next higher or lower range is selected, the reading increases or decreases by a decade (factor of 10).

A parameter value is keyed in by placing the cursor on the digit to be changed and using one of the following methods:

- Use the EDIT ▲ or ▼ or the RANGE ▲ or ▼ keys to increment or decrement the digit.
- Use the number keys (0 through 9) to key in the value at the selected digit.
- Use the ± key to change source value polarity, regardless of cursor position.

*NOTE   You can clear a parameter value by placing the cursor on it and pressing the **0000** (MENU) key.*

- Boolean selections (such as ON/OFF and HIGH/LOW) are toggled by placing the cursor on the selection and pressing an EDIT or a RANGE ▲ or ▼ key.

- A change is only executed when ENTER is pressed. Entering an invalid parameter generates an error, and the entry is ignored. However, entering an out-of-range value (too small or too large) selects the lower or upper limit, respectively.
- Press the **EXIT** key to return to the previous menu. Any change that is not entered is cancelled when EXIT is pressed.

NOTE *Although changes to menu items require that ENTER key be pressed, changes to source values will be updated immediately.*

Figure 1-3
Main menu tree

Press the **MENU** key (Use ◀ and ▶ to select an item, and then press the **ENTER** Key)
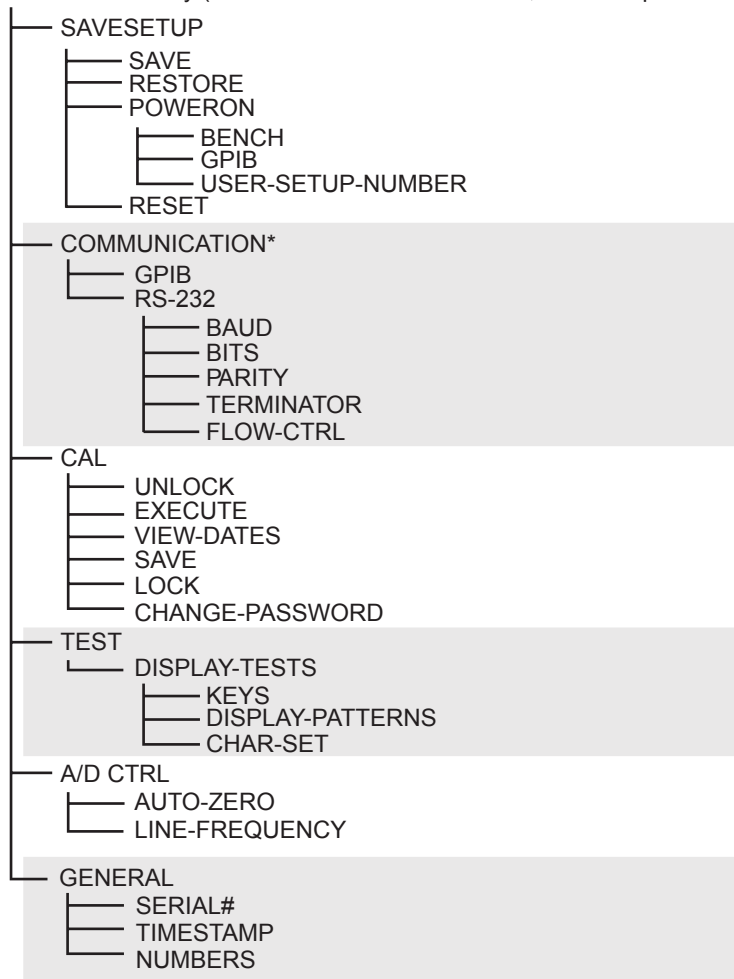
```
├── SAVESETUP
│     ├── SAVE
│     ├── RESTORE
│     ├── POWERON
│     │      ├── BENCH
│     │      ├── GPIB
│     │      └── USER-SETUP-NUMBER
│     └── RESET
├── COMMUNICATION*
│     ├── GPIB
│     └── RS-232
│            ├── BAUD
│            ├── BITS
│            ├── PARITY
│            ├── TERMINATOR
│            └── FLOW-CTRL
├── CAL
│     ├── UNLOCK
│     ├── EXECUTE
│     ├── VIEW-DATES
│     ├── SAVE
│     ├── LOCK
│     └── CHANGE-PASSWORD
├── TEST
│     └── DISPLAY-TESTS
│            ├── KEYS
│            ├── DISPLAY-PATTERNS
│            └── CHAR-SET
├── A/D CTRL
│     ├── AUTO-ZERO
│     └── LINE-FREQUENCY
└── GENERAL
      ├── SERIAL#
      ├── TIMESTAMP
      └── NUMBERS
```

*   When changing interface selections, the Model 6482 performs a power-on reset. To check or change options of the selected interface, you must re-enter the menu structure.

Editing voltage bias values

Use the following keys to edit voltage bias values:

- SRC1 or SRC2: selects the channel 1 (Src1) or channel 2 (Src2) source display field on the lower line for editing. A blinking cursor will appear in the field to be edited. If no key is pressed within a few seconds, the edit mode will be cancelled automatically.
- EDIT ◀ and ▶ places the display cursor on the display digit to be changed.

- EDIT ▲ or ▼ increments or decrements the source value. Note that pressing either of these keys will automatically enable the source edit mode.
- RANGE ▲ or ▼ selects the voltage bias source range (10 V or 30 V).
- Numeric keys (0 through 9) allow you to directly enter source values.
- EXIT exits the edit mode without waiting for the time-out period.

The basic procedure for editing source values is outlined below. See Section 3, "Basic measurement procedure," for more details.

1. Press the **SRC1** or **SRC2** key so that the blinking cursor is in either the channel 1 (Src1) or channel 2 (Src2) source display field to be edited.
2. You can use the RANGE ▲ and ▼ keys to select the source range. Use the lowest range possible for best accuracy.
3. To increment or decrement the displayed source value, use the EDIT ◄ and ► keys to place the blinking cursor on the digit to be changed, then increment or decrement the value with the EDIT ▲ and ▼ keys. Note that the source value will be updated immediately; you do not need to press the ENTER key to complete the process.
4. To enter the source value directly, enter in the value with the numeric keys while the cursor is blinking. The source value will be updated immediately.

Configuration menus

There are a number of configuration menus that can be accessed by pressing the **CONFIG** key followed by the appropriate function or mode key. For example, you can configure the measurement functions by pressing the **CONFIG** key, and then pressing the **MSR1** (channel 1) or **MSR2** (channel 2) key. Configuration menus, which are summarized in Table 1-4 through Table 1-10, are available for the following operating modes:

- Measurement functions (MSR1, MSR2, RATIO, DELTA); see Table 1-4.
- Voltage source (SRC1 and SRC2); see Table 1-5.
- REL, FILTER, and RANGE; see Table 1-6.
- LIMIT; see Table 1-7.
- TRIG; see Table 1-8.
- SWEEP, DIGITS, SPEED, and STORE; see Table 1-9.
- ON/OFF OUTPUT and DISPLAY TOGGLE; see Table 1-10.

These configuration menus are covered in detail in the applicable sections of this manual.

Table 1-4
Measurement configuration menus

Configuration menu item	Description
CONFIG > **MSR1**	Configure channel 1 measurement.
CONFIG MSR1 BUTTON	
I	Current measurement.
I/V	Conductance (I/V) measurement.
V/I	Resistance (V/I) measurement.
MX + B_UNIT	MX + B measurement, program M and B values.
P⚡	Electrical power (V × I) measurement.
CONFIG > **MSR2**	Configure channel 2 measurement.
CONFIG MSR2 BUTTON	
I	Current measurement.
I/V	Conductance (I/V) measurement.
V/I	Resistance (V/I) measurement.
MX + B_UNIT	MX + B measurement, program M and B values.
P⚡	Electrical power (V × I) measurement.
CONFIG > **RATIO**	Configure ratio function.
CONFIG RATIO BUTTON	
MSR1/MSR2	MSR1/MSR2 function.
MSR2/MSR1	MSR2/MSR1 function.
CONFIG > **DELTA**	Configure delta function.
CONFIG DELTA BUTTON	
MSR1-MSR2	MSR1-MSR2 function.
MSR2-MSR1	MSR2-MSR1 function.

Table 1-5
Source configuration menus

Configuration menu item	Description
CONFIG > **SRC1**	Configure voltage source.
CONFIG SRC1	Configure delay and ground connect mode.
DELAY	Set source delay (0 to 9999.998 s).
AUTO-DELAY	Control source autodelay.
DISABLE	Disable autodelay.
ENABLE	Enable autodelay (delay depends on current range).
GND-CONNECT	Control ground connect mode.
DISABLE	Disable ground connect mode.
ENABLE	Enable ground connect mode.
CONFIG > **SRC2**	Configure voltage source.
CONFIG SRC2	Configure delay and ground connect mode.
DELAY	Set source delay (0 to 9999.998 s).
AUTO-DELAY	Control source autodelay.
DISABLE	Disable autodelay.
ENABLE	Enable autodelay (delay depends on current range).
GND-CONNECT	Control ground connect mode.
DISABLE	Disable ground connect mode.
ENABLE	Enable ground connect mode.

Table 1-6
Rel, filter, and range configuration menus

Configuration menu item	Description
CONFIG > **REL**	Program REL value.
CONFIG > **FILTER**	Configure filter.
CONFIGURE FILTERING	
AVERAGE-MODE	Select average mode filter.
MOVING	Moving average.
REPEAT	Repeat average.
AVERAGE-COUNT	Set average filter count (1 to 100).
ADVANCED	Enable or disable advanced filter.
DISABLE	Disable advanced filter.
ENABLE	Enable advance filter, set noise tolerance.
MEDIAN-RANK	Select median rank (0 to 5).
CONFIG > **RANGE** ▲	Program upper range limit.
CONFIG > **RANGE** ▼	Program lower range limit.

Table 1-7
Limit configuration menu

Configuration menu item	Description
CONFIG > **LIMIT**	Configure limit tests.
CONFIG LIMITS MENU	
H/W-LIMITS	Control and set fail mode for Limit 1, Limit 2 tests.
LIMITS1/2	
CONTROL	Control Limit 1 or 2 test.
DISABLE	Disable Limit 1 or 2 test.
ENABLE	Enable Limit 1 or 2 test.
FAIL-MODE	Select Limit 1 or 2 fail mode.
IN	Fail when in compliance.
OUT	Fail when out of compliance.
S/W-LIMITS	Control Limit 3 to Limit 6 tests.
LIMITS3/4/5/6	
CONTROL	Enable or disable Limit 3 to 6 tests.
DISABLE	Disable test.
ENABLE	Enable test.
LOLIM	Set low limit.
HILIM	Set high limit.
FEED	Select input path (MSR1, MSR2, RATIO, or DELTA).

Table 1-8
Trigger configuration menu

Configuration menu item	Description
CONFIG > TRIG	Configure triggering.
CONFIGURE TRIGGER	
ARM LAYER	Configure trigger model arm layer.
ARM-IN	Select arm layer detection event.
IMMEDIATE	Immediate event detection.
GPIB	GPIB GET or *TRG.
TIMER	After timer interval elapses, enter interval.
MANUAL	Front-panel TRIG key.
TLINK	Enter TLINK line and state.
ONCE	Bypass event detection once.
NEVER	Never bypass event detection.
↓STEST	When SOT is pulsed low.
ONCE	Bypass event detection once.
NEVER	Never bypass event detection.
↑STEST	When SOT line is pulsed high.
ONCE	Bypass event detection once.
NEVER	Never bypass event detection.
↑↓STEST	When SOT line is pulsed low or high.
ONCE	Bypass event detection once.
NEVER	Never bypass event detection.
ARM-OUT	Configure arm layer output trigger.
LINE	Select trigger link output line (1 through 6).
EVENTS	Enable or disable events.
TRIG-LAYER-DONE	Enable (ON) or disable (OFF) on exiting trigger layer.
COUNT	Specify arm count.
FINITE	Programmable count.
INFINITE	Continuous count.
TRIG-LAYER	Configure trigger layer of trigger model.
TRIGGER-IN	Select trigger layer detection event.
IMMEDIATE	Trigger event occurs immediately.
TRIGGER-LINK	Select trigger link line as event (1 through 6).
ONCE	Bypass event detection once.
NEVER	Never bypass event detection.
TRIGGER-OUT	Configure trigger layer output trigger.
LINE	Select trigger link line (1 through 6).
EVENTS	Enable (ON) or disable (OFF)*
DELAY	Program trigger delay time.
COUNT	Program trigger count.
HALT	Return instrument to idle state.

*For source (SOURCE), delay (DELAY), and measure (MEAS).

Table 1-9
Sweep, digits, speed, and data store configuration menus

Configuration menu item	Description
CONFIG > SWEEP	Configure sweeps of the selected channel*
CONFIGURE SWEEPS	
TYPE	Select sweep type.
NONE	Disable sweep.
STAIR	Staircase sweep, program START, STOP, STEP.
LOG	Logarithmic sweep, program START, STOP, # POINTS.
CUSTOM	Custom sweep, program parameters.
# POINTS	Program number of sweep points.
ADJUST-POINTS	Set individual point values.
INIT	Set first point value.
SWEEP-COUNT	Set sweep count.
FINITE	Program sweep count value.
INFINITE	Continuous sweep.
SOURCE-RANGING	Set sweep ranging mode.
BEST-FIXED	Best fixed range based on maximum value.
AUTO-RANGE	Autorange during sweep.
FIXED	Set fixed source range.
CONFIG > DIGITS	Set display number of digits for both channels.
DISPLAY DIGITS	Select 3.5, 4.5, 5.5, or 6.5.
CONFIG > SPEED	Set measurement speed for both channels.
SPEED-ACCURACY-MENU	
FAST	Fast speed.
MED	Medium speed.
NORMAL	Normal speed.
HI ACCURACY	Maximum accuracy.
OTHER	Program NPLCs (number of power line cycles).
CONFIG > STORE	Configure data store timestamp.
STORE TIMESTAMP	
ABSOLUTE	Absolute timestamp.
DELTA	Delta timestamp.

* To select a channel , press the CHANNEL SELECT key.

Table 1-10
Output and display configuration menus

Configuration menu item	Description
CONFIG > ON/OFF OUTPUT	Configure output.
CONFIG OUTPUT	
AUTO OFF	Enable or disable auto-off mode.
DISABLE	Keep output on.
ENABLE	Turn output off after each measurement.
ALWAYS	After each measurement.
AFTER-TRIG-COUNT	When trigger count expires.
ENABLE	Enable or disable output enable.
DISABLE	Disable output enable.
ENABLE	Enable output enable.
CONFIG > DISPLAY TOGGLE	Enable or disable display.
DISABLE DISPLAY	
NOW	Disable display immediately.
NEVER	Never disable display.
SWEEP	Turn display off during sweep.
STORE	Turn display off during buffer store.
CONFIG > DISPLAY CHANNEL SELECT	Configure displayed channel.
CONFIGURE DISPLAY	
NOW	Select channel displayed immediately.
NEVER	Never select displayed channel.
SWEEP	Select channel displayed during sweep.
STORE	Select channel displayed during buffer store.

This page left blank intentionally.

Connections

In this section:

Connection precautions

The following paragraphs summarize precautions that should be observed when making test connections to the Model 6482.

WARNING   *Maximum common-mode voltage (voltage between INPUT analog common and chassis ground) is 200 V. Exceeding this value may result in a shock hazard.*

When making connections, do not leave any exposed connections. Be sure that all external circuits are properly insulated.

CAUTION   **Maximum INPUT and OUTPUT connector rating is 30 V at 20 mA. Exceeding these values may result in instrument damage.**

Input and output connectors

Figure 2-1 shows the location of the input and output connectors on the rear panel. These connectors have the following functions.

- **Input connectors**
  INPUT CHANNEL 1 and CHANNEL 2: A 3-lug triaxial connector for direct current input from the device under test (DUT) to channel 1 (or channel 2).
- **Analog output connectors**
  ANALOG OUT CHANNEL 1 and CHANNEL 2: A 3-lug triaxial connector that provides a -10 V to +10 V DC output analogous to the input current applied to channel 1 or channel 2.
- **Voltage source (output) connectors**
  VOLTAGE SOURCE OUT CHANNEL 1 or CHANNEL 2: A safety banana jack is used to apply the channel 1 HI or channel 2 HI voltage bias source signal to the DUT.

Figure 2-1
Model 6482 rear panel showing input and output connectors
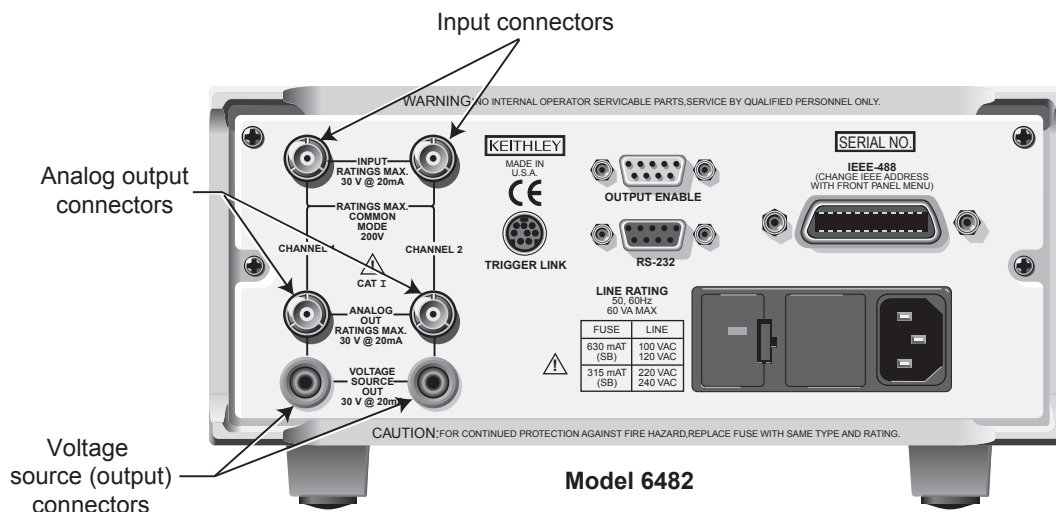
Connector terminals

The electrical configuration of the input, analog output, and voltage source connectors are shown in Figure 2-2 and Table 2-1.

Figure 2-2
Connector terminals
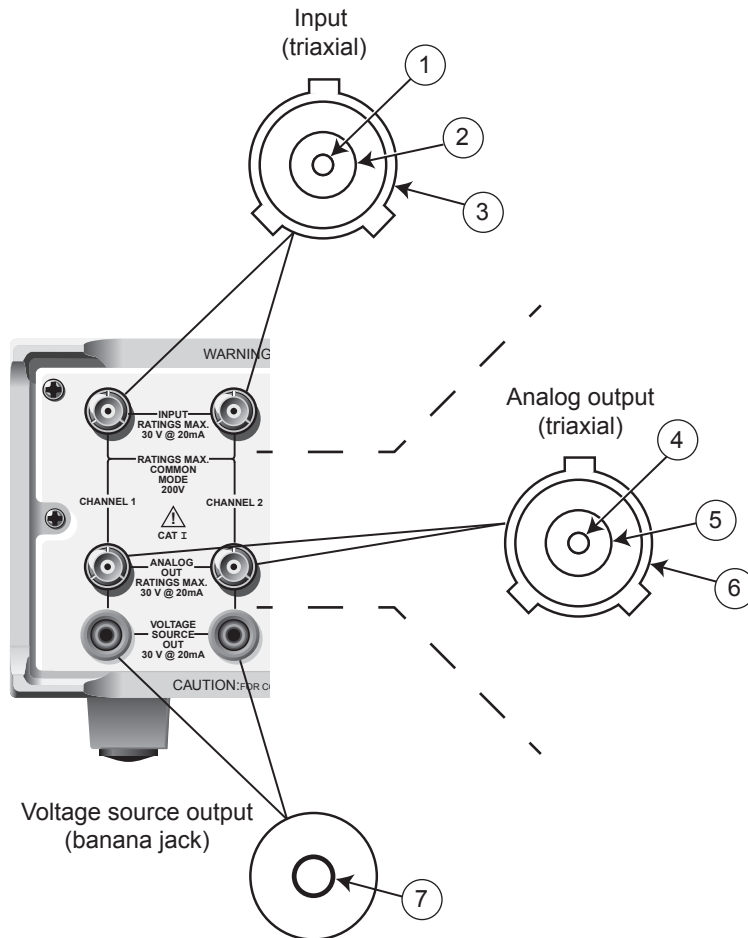
Table 2-1
Connector terminal legend

Item	Connector	Description	Terminal
1		Input HI	Inner conductor
2	Input	Input LO (analog common)	Inner shield
3		Chassis ground	Outer shield
4		Analog output HI	Inner conductor
5	Analog output	Analog output LO (analog common)	Inner shield
6		Chassis ground	Outer shield
7	Voltage source output	Voltage source (output)	Safety banana jack

Input connectors

The electrical configuration of each triaxial INPUT connector is shown in Figure 2-2. Connector terminals are designated as follows:

- Inner (center) conductor of the connector (and triaxial cable): Input HI. This terminal connects to one terminal of the device under test (DUT).
- Inner shield (ring) of the connector and inner cable shield: Input LO (analog common).
- Outer shield (ring) of connector (shell) and outer cable shield: Chassis ground.

Refer to Current measurement connections for additional information on using the input connectors.

Analog output connectors

WARNING   ***Analog output low can float up to ±30 V. Use care to avoid a shock hazard when using the analog outputs.***

Figure 2-2 shows the terminal configuration of each triaxial ANALOG OUT connector. Output terminals are as follows:

- Inner (center) conductor: Analog output HI
- Inner shield: Analog output LO (floating analog common)
- Outer shield: Chassis ground

Refer to Analog output connections for additional information on using the analog output connectors.

Voltage source output connectors

Each channel has a single OUTPUT banana jack for the voltage bias source. Each connector is the HI terminal for the corresponding voltage source channel and connects to one terminal of the device under test (DUT).

NOTE   *There are no separate external connections for voltage source LO. The LO node of each voltage bias source is internally connected to the floating analog common. See "Connection methods" later in this section for connection details.*

Refer to Voltage source connections for additional information on using the voltage source output connectors.

Output enable

An interlock switch can be used with the Model 6482 output enable line on the OUTPUT ENABLE connector to inhibit the voltage bias source outputs. When output enable is activated, the Model 6482 outputs will turn off when the lid of the test fixture is opened.

WARNING   ***To prevent electric shock, test connections must be configured such that the user cannot come in contact with conductors or any device under test (DUT) that is in contact with the conductors. Safe installation requires proper shields, barriers, and grounding to prevent contact with conductors. You must always assume that power is present until you verify that the Model 6482 output is off. Operator protection and safety are the responsibility of the person installing the product.***

See Section 11 for complete details on using the output enable circuit.

Connection methods

The Model 6482 can be used as an ammeter to make current measurements, or as a voltage source to provide bias voltage to any suitable device.

Current measurement connections

Figure 2-3 shows typical connections using channel 1 to measure the device under test (DUT) current. Note that the INPUT HI terminal (center conductor) is connected to DUT HI, while the INPUT LO terminal (analog common) is connected to DUT LO.

NOTE   *The DUT can be biased and measured using only a triaxial cable with the ground connect mode. For more information, see Ground connect mode connections.*

Figure 2-3
Current measurement connections
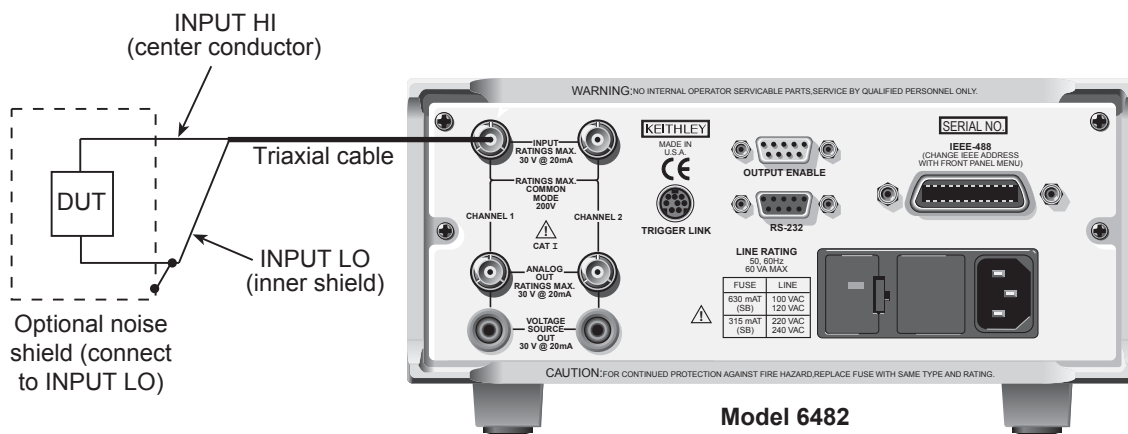
Voltage source connections

Figure 2-4 shows the connecting scheme for using the Model 6482 as a voltage bias source. Note that the device under test (DUT) HI terminal is connected to the OUTPUT terminal of the voltage source (Channel 1 output), and the DUT LO terminal is connected to INPUT LO (analog common, which is the inner shield of the INPUT jack). The maximum voltage source output is 30 V at 20 mA.

Figure 2-4
Voltage source connections
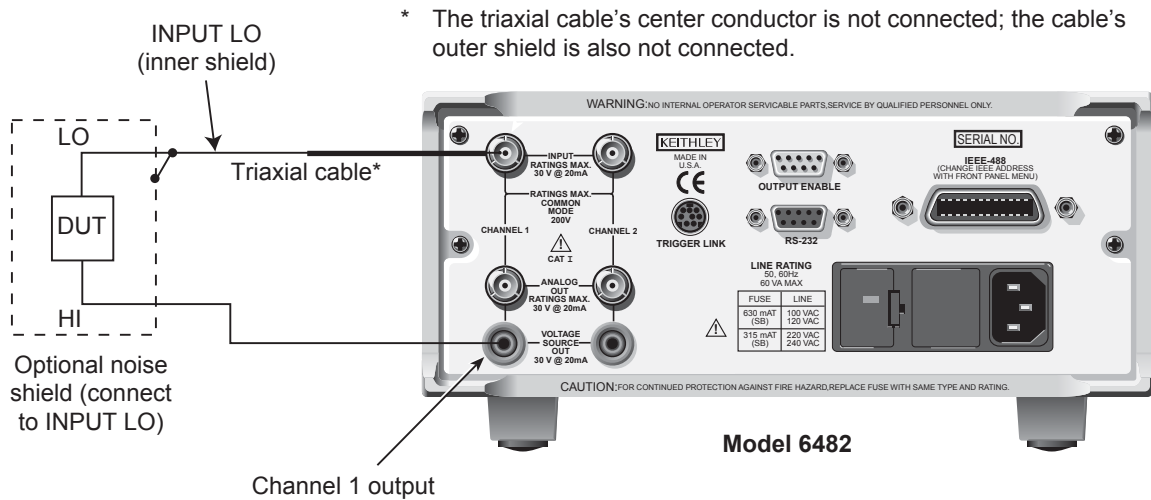
> **NOTE**   *If an optional noise shield is used, connect the shield to INPUT LO (inner shield of triaxial cable).*

Connection considerations

To avoid noise and offset currents that could degrade measurement accuracy, be sure to use only quality, low-noise triaxial cables for INPUT connections. Also, keep cables and test fixtures away from vibration and varying temperatures to minimize generated cable currents. See Appendix D, "Measurement Considerations," for information on these and other possible measurement problems and how to avoid them.

Ground connect mode connections

The VOLTAGE SOURCE output jack for either channel can either be connected to chassis ground or left floating, depending on the selected ground connect mode. See Section 3, "Ground connect mode," for details. With ground connect disabled, you must make separate connections to the VOLTAGE SOURCE OUTPUT jack, as shown in Figure 2-4. However, with ground connect enabled, you can bias and measure the device under test (DUT) using a single triaxial cable, as shown in Figure 2-5. Note that one device under test (DUT) terminal is connected to INPUT HI, while the other DUT terminal is connected to chassis ground through the outer shield of the triaxial cable connection. Figure 2-6 shows an equivalent circuit.

Figure 2-5
Test connections using ground connect mode
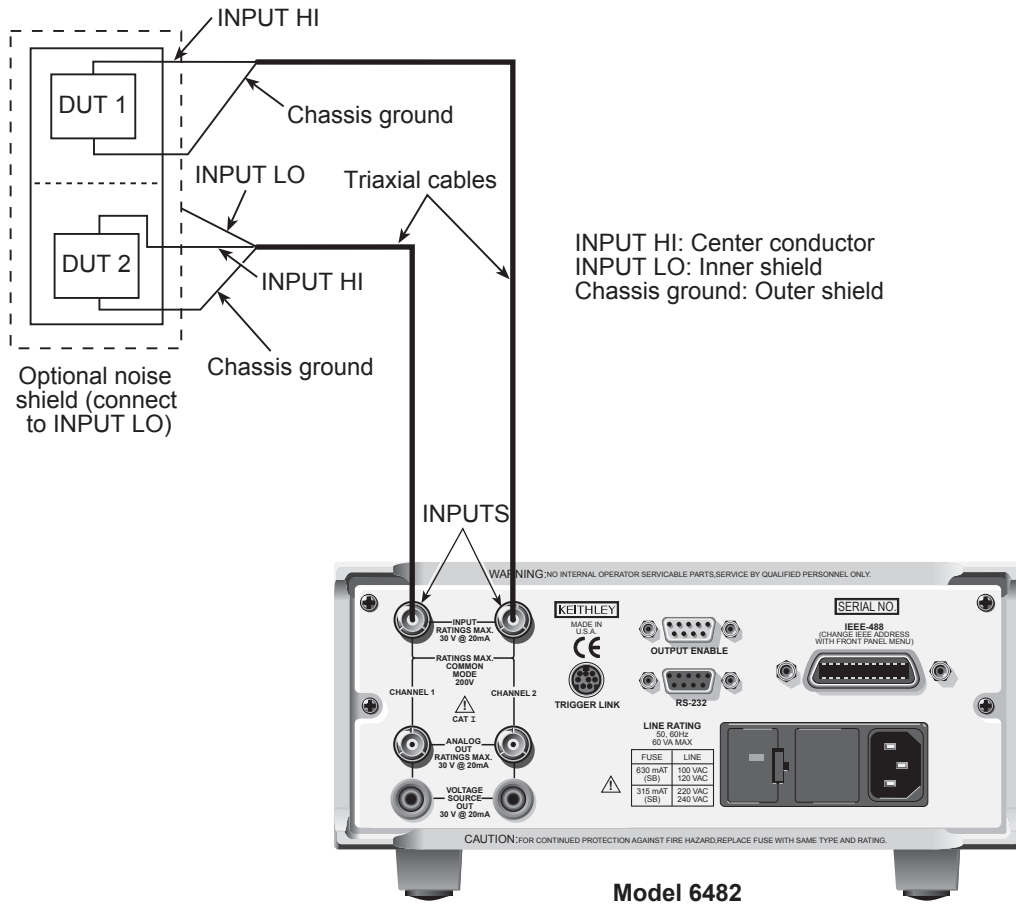
Figure 2-6
Ground connect mode equivalent circuit

Model 6482
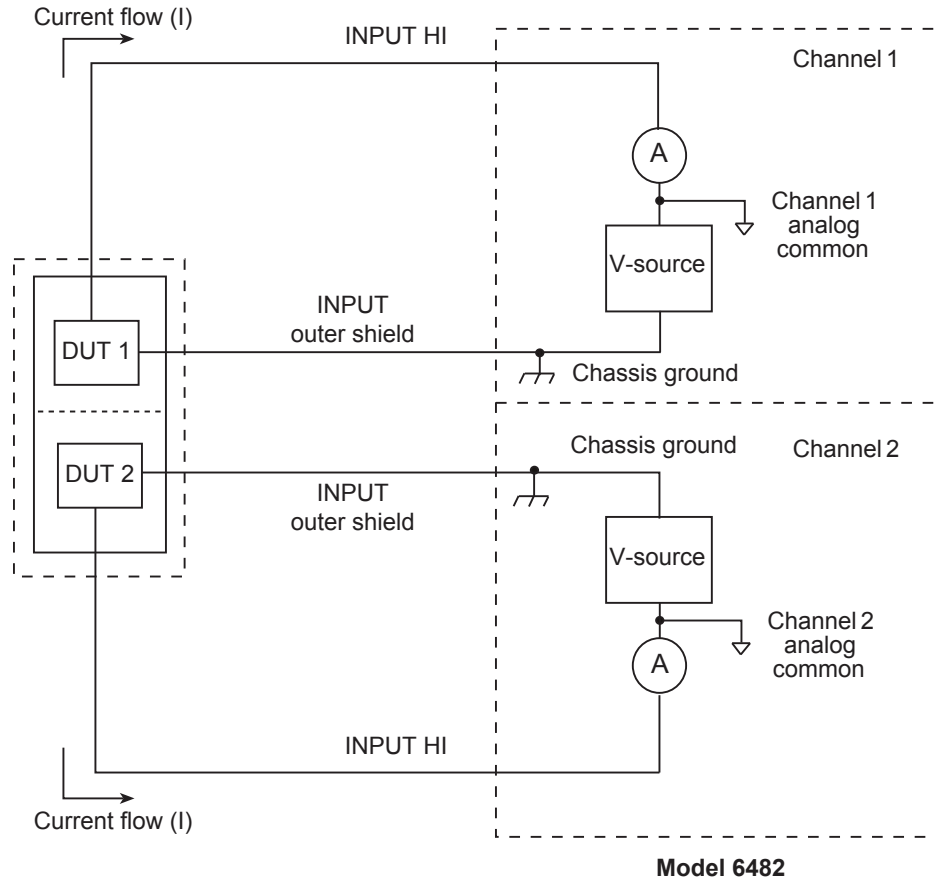
Analog output connections

> *WARNING*   ***Analog output low can float up to ±30 V. Use care to avoid a shock hazard when using the analog outputs.***

For connector location, refer to "Input and output connectors" on page 2-2; for terminal information, refer to "Connector terminals" on page 2-3.

Nonisolated connections

Figure 2-7 shows typical nonisolated analog output connections. Note that analog output HI (center conductor) is connected to INPUT HI of the measurement instrument and analog output LO (inner shield) is connected to INPUT LO.

CAUTION   **With the ground connect mode enabled, analog output LO can float up to ±30 V above chassis ground, depending on the voltage bias source setting. Connecting the analog outputs to external equipment that does not allow analog output low to float up to ±30 V may result in damage to the Model 6482 voltage bias sources when the ground connect mode is enabled.**

Figure 2-7
Nonisolated analog output connections
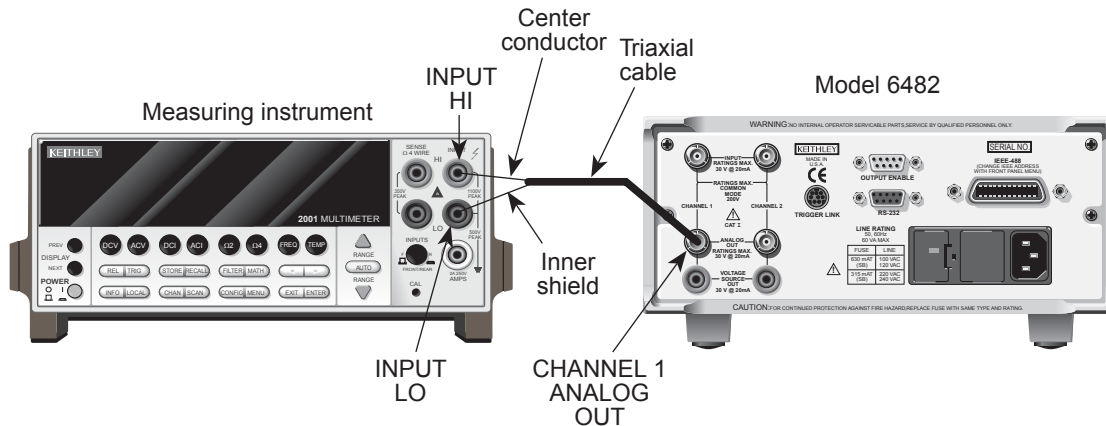
Isolated connections

To maintain proper isolation between the analog outputs and chassis ground, use the isolated connections shown in Figure 2-8. A wide-bandwidth single channel signal conditioning module (Analog Devices 3B41-03 Isolated Wideband Voltage Input subsystem) is used to provide isolation between the Model 6482 ANALOG OUT and the measuring instrument. Analog output HI (center conductor) is connected to the HI input terminal of the isolator; analog output LO (inner shield) is connected to the LO input. As shown in the figure, the two isolator V_{OUT} terminals are connected to the measuring instrument HI and LO INPUT terminals.

Refer to the documentation provided with the signal conditioning module for connections, power supply, and configuration information for your application.

Figure 2-8
Isolated analog output connections
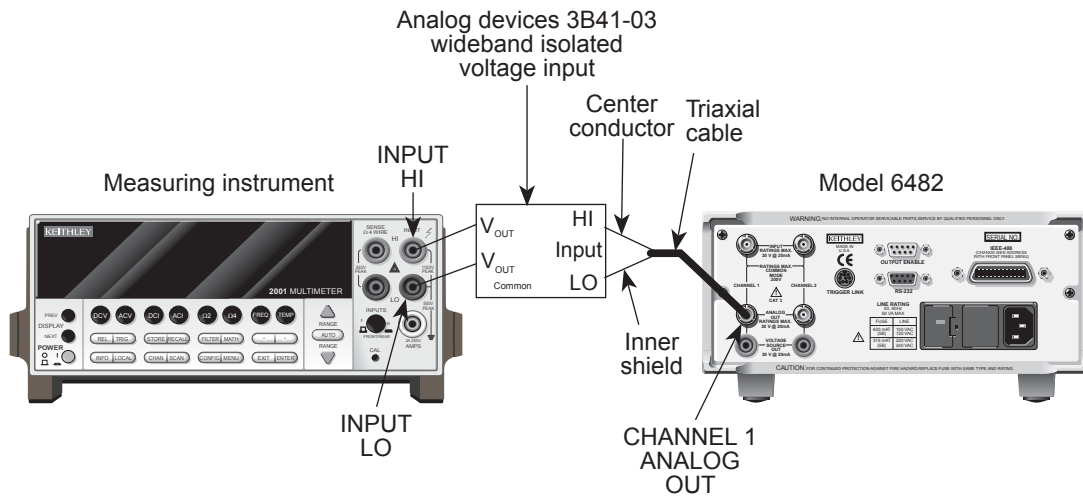
Equivalent circuits

Figure 2-9 shows an equivalent circuit of the analog outputs with the ground connect mode disabled. Only one channel is shown; the other channel is identical. Note that the analog output circuit consists of a gain/buffer amplifier, and the analog output LO is connected to floating common.

Figure 2-10 shows an equivalent circuit with the ground connect mode enabled. In the figure, because one side of the voltage bias source is connected to chassis ground, analog output LO is allowed to float up to ±30 V above chassis ground (dependent on the voltage bias source setting).

Figure 2-9
Analog output circuit with ground connect disabled
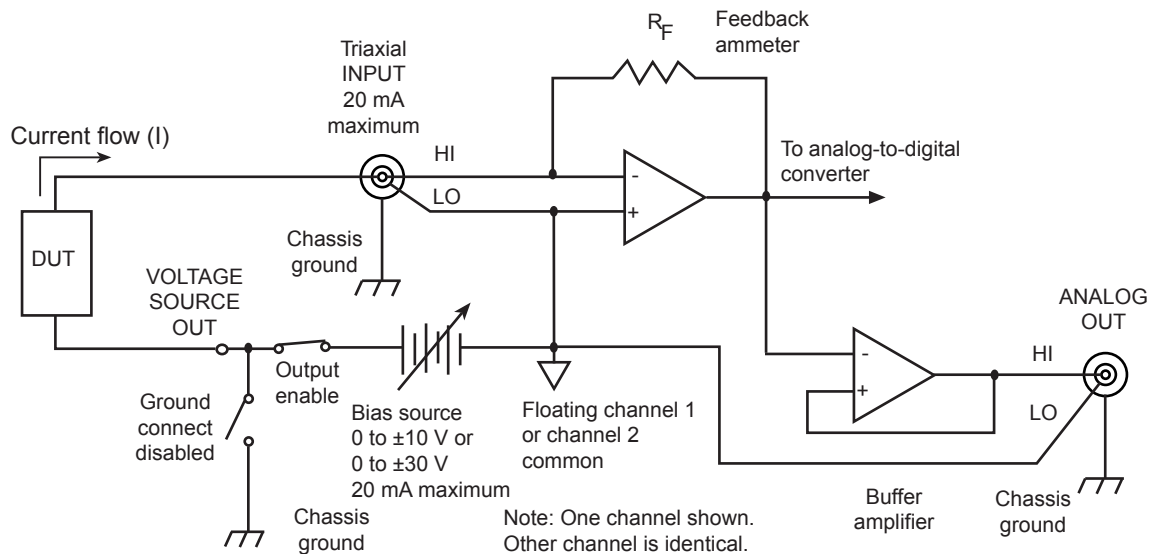
Figure 2-10
Analog output circuit with ground connect enabled
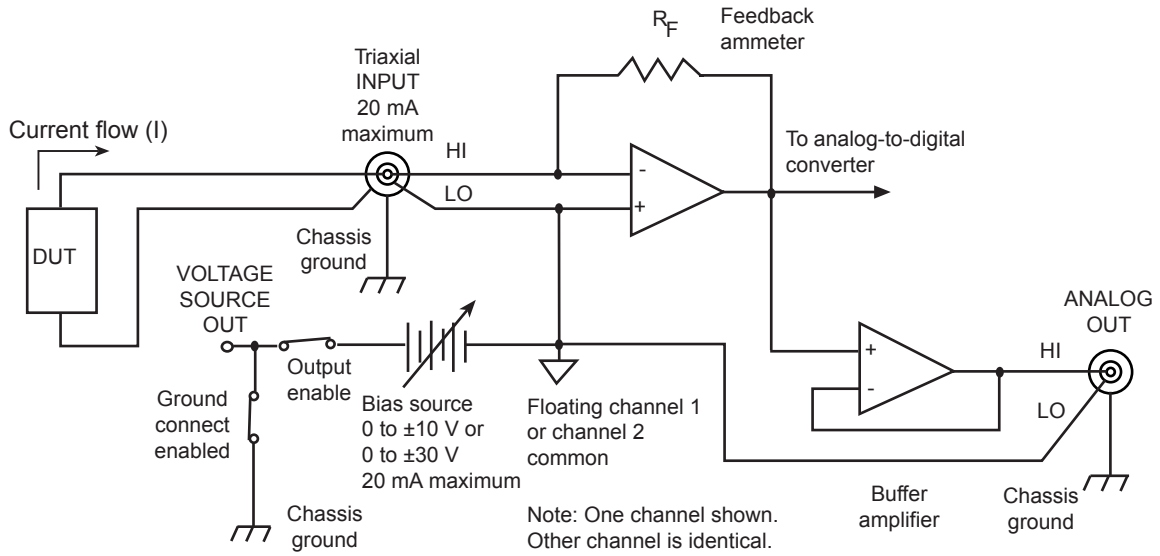
This page left blank intentionally.

Basic Operation

In this section:

Operation overview

This topic discusses current measurement and voltage bias source capabilities, ranges, compliance, and fundamental measurement and voltage bias circuit configuration.

Measurement and voltage bias capabilities

The Model 6482 has the following measurement and voltage bias capabilities:

- **Measure current**: Each Model 6482 channel can measure DC currents from ±1 fA to ±20 mA. Each channel has eight current ranges: 2 nA, 20 nA, 200 nA, 2 μA, 20 μA, 200 μA, 2 mA, and 20 mA.
- **Source voltage**: Each Model 6482 channel can output DC voltage from ±500 μV to ±30 V. Each channel has two ranges: 10 V and 30 V.

NOTE   Both channel 1 and channel 2 are floating and independent of one another.

Ranges

Table 3-1 summarizes current measurement ranges, resolutions, and maximum readings while Table 3-2 lists voltage bias source ranges, resolutions, and maximum voltage and current. See Section 4 for more details on ranging.

Table 3-1
Current measurement ranges

Model 6482 I-range	Maximum resolution	Maximum reading
2 nA	1 fA	±2.1 nA
20 nA	10 fA	±21 nA
200 nA	100 fA	±210 nA
2 μA	1 pA	±2.1 μA
20 μA	10 pA	±21μA
200 μA	100 pA	±210μA
2 mA	1 nA	±2.1 mA
20 mA	10 nA	±21 mA

Table 3-2
Voltage source ranges

Model 6482 V-range	Resolution	Maximum voltage	Maximum current
10 V	400 μV	±10 V	20 mA
30 V	4 mV	±30 V	20 mA

Compliance

Each voltage bias source has a maximum output level of 30 V at 20 mA. If the load resistance is low enough so that the instrument exceeds the 20 mA current limit, the instrument is considered to be in compliance, and the corresponding source display field (Src1 or Src2) will flash. See Section 9, "Bias source operating boundaries," for more information. You can also use compliance for limit testing (see Section 8), and the instrument can be programmed over the GPIB (see Section 13) to generate an SRQ under compliance conditions.

Basic circuit configuration

The fundamental circuit configuration for the Model 6482 is shown in Figure 3-1. Note that the instrument has two separate channels (one channel shown), each of which includes a feedback ammeter and a 0 ±10 V or 0 ±30 V voltage bias source. Figure 3-1 also shows the output enable circuit, which is shown enabled (see Section 11), and the ground connect relay (see "Ground connect mode"). Note that the device under test (DUT) is connected between the triaxial INPUT jack and the SOURCE OUTPUT terminal.

Figure 3-1
Basic circuit configuration
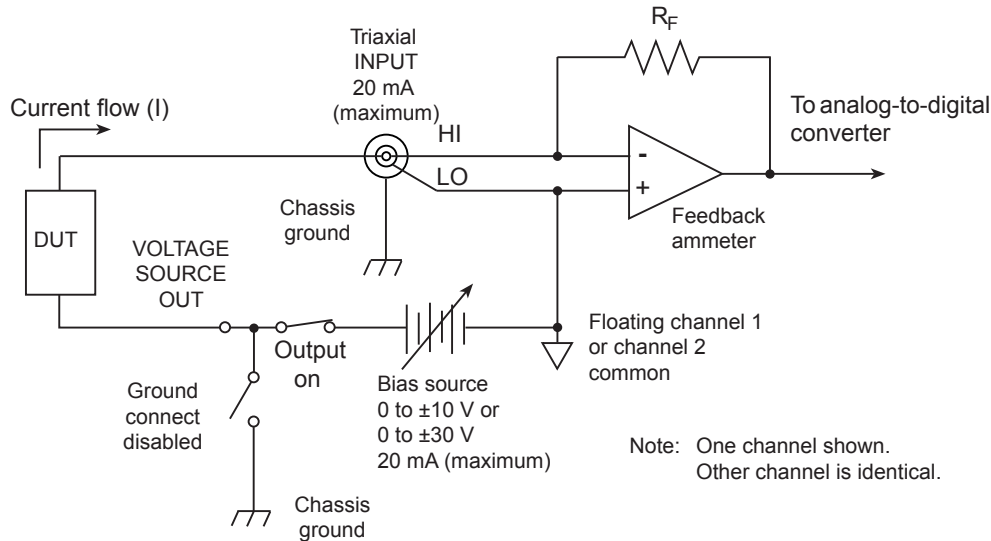
Operation considerations

The following paragraphs discuss warm-up period, autozero, and source delay.

Warm-up

The Model 6482 must be turned on and allowed to warm up for at least one hour to achieve rated accuracies.

Autozero

Every analog-to-digital conversion (current reading) is calculated from a series of zero, reference, and signal measurements. With autozero enabled, all three of these measurements are performed for each reading to achieve rated accuracy. With autozero disabled, zero and reference are not measured. Disabling autozero increases measurement speed, but zero drift will eventually degrade accuracy. With autozero disabled, periodically change measurement speed.

Temperature changes across components within the instrument can cause the reference and zero values for the analog-to-digital converter to drift due to thermo-electric effects. Autozero acts to negate the effects of drift in order to maintain measurement accuracy over time. Without autozero enabled, current measurements can drift and become erroneous.

NOTE   *Autozero control affects both channels simultaneously.*

Front-panel autozero

Set the autozero state from the front panel as follows:

1. Press the **MENU** key.
2. Select **A/D-CTRL** from the main menu, and then press the **ENTER** key.
3. Select **AUTO-ZERO**, and then press the **ENTER** key.
4. Select ENABLE or DISABLE as appropriate, and then press the **ENTER** key.
5. Press the **EXIT** key as necessary to return to normal display.

Remote command autozero

Use the :SYSTem:AZERo command to enable or disable autozero over a remote interface. For example, send the following command to disable autozero:

```
:SYST:AZER ON
```

Conversely, send this command to disable autozero:

```
:SYST:AZER OFF
```

Source delay

The source delay options are used to set the settling time for the voltage bias source. This source delay is the delay phase of the source-delay-measure (SDM) cycle. See Section 9, "Measurement Concepts," for more information.

Autodelay period

The autodelay period depends both on the current range (Table 3-3) and, during sweeps (Section 7), on the voltage step size. With the source autodelay mode enabled, the total source delay for both channels will be the sum of the current-range-based delay in Table 3-3 and the voltage slew time for the channel that has the greater total delay. Worst-case voltage slew time, T_{VS}, is given by the equation:

$$T_{VS} = (3.07ms)/V$$

Table 3-3
Automatic source delay

I-range	I-range-based autodelay
2 nA	16 ms
20 nA	16 ms
200 nA	12.25 ms
2 µA	12.25 ms
20 µA	3.55 ms
200 µA	3.55 ms
2 mA	530 µs
20 mA	530 µs

For example, with the 2µA range and a voltage step size of 0.5 V, the total delay is:

Delay = 12.25 ms + (0.5 $\times$ 3.07 ms)
Delay = 13.785 ms

Output slew time

Figure 3-2 demonstrates the effect of output slew time on overall settling. Total time after changing the source value before accurate measurements can be taken includes both the slew time and the measure circuit settling time.

NOTE   *When not performing a sweep, the output slew time is not part of the automatic source delay. Therefore, for best accuracy when turning the output on or changing the voltage source value, allow a delay of 3.07 ms/V (milliseconds per volt) of output change before triggering a reading. This delay can be programmed by setting the trigger delay (see Section 8, "Front-panel operation of the trigger model"). Also, while the output is slewing, the compliance status bit for the channel will be set (Section 13, "Status Structure"). You can poll the compliance bit to determine when the programmed source value is reached and an accurate reading can be taken.*

Figure 3-2
Output slew time
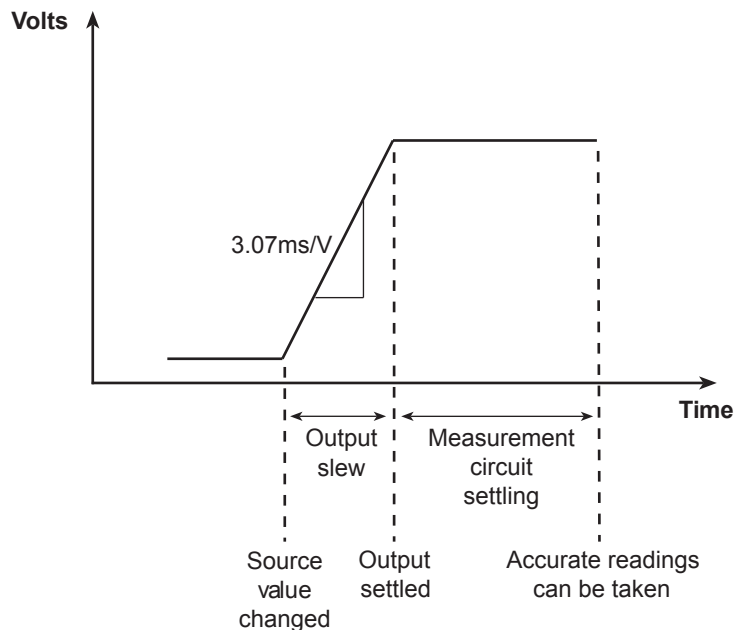
Manual delay

Manual delay for each channel can be set from 0000.00000 to 9999.99800 seconds. Manually setting the delay disables autodelay.

Front panel source delay

NOTE   *The delay period is the same for both sources. If you change the delay on one channel, the delay period on the other source will change to the same value.*

To set the manual source delay from the front panel:

1. Press the **CONFIG** key, and then press the **SRC1** or **SRC2** key.
2. Select **DELAY** from the displayed choices, and then press the **ENTER** key.
3. Enter your delay value, and then press the **ENTER** key.

To set the automatic source delay state from the front panel:

1. Press the **CONFIG** key, and then the **SRC1** or **SRC2** key.

2. Select **AUTO-DELAY** from the displayed choices, and then press the **ENTER** key.

3. Select **ENABLE** or **DISABLE** as appropriate, and then press the **ENTER** key.

Remote command source delay

Use the appropriate command shown in Table 3-4 to program the source delay over a remote interface. See Section 16 for details. For example, the following command sets the source delay to 50 ms:

```
:SOUR1:DEL 0.05
```

Table 3-4
Source delay commands

Command	Description
:SOURce[1]:DELay <Delay> :SOURce[1]:DELay:AUTO <State> :SOURce2:DELay <Delay> :SOURce2:DELay:AUTO <State>	Set source delay. Enable or disable autodelay (ON or OFF). Set source delay. Enable or disable autodelay (ON or OFF).

Ground connect mode

The VOLTAGE SOURCE output jack for either channel can be left floating or connected to chassis ground by selecting the appropriate ground connect mode. When ground connect is enabled, the VOLTAGE SOURCE terminal will be connected to chassis ground, as shown in Figure 3-3. This configuration allows you to bias and measure the device under test (DUT) using a single triaxial cable as shown. When ground connect is disabled (Figure 3-4), you must make separate SOURCE OUTPUT connections to the DUT. See Section 2 for connection details.

NOTE   The ground connect mode can be individually controlled for each channel.

Front panel ground connect

Enable or disable the ground connect mode as follows:

1. Press the **CONFIG** key, and then press the **SRC1** or **SRC2** key.

2. Select **GND-CONNECT**, and then press the **ENTER** key.

3. Choose **ENABLE** or **DISABLE** as appropriate, and then press the **ENTER** key.

Remote command ground connect

Use the appropriate command shown in Table 3-5 to program the ground connect mode over a remote interface (see Section 16 for details). For example, the following command enables ground connect on channel 1:

```
:SOUR1:GCON ON
```

Table 3-5
Ground connect commands

Command	Description
:SOURce[1]:GCONnect <State> :SOURce2:GCONnect <State>	Enable or disable channel 1 ground connect (ON or OFF). Enable or disable channel 2 ground connect (ON or OFF).

Figure 3-3
Ground connect enabled

Triaxial
INPUT
HI   To ammeter
LO
Chassis
ground
SOURCE
OUTPUT
Output
ON
Bias source
Floating
common
Ground connect
relay closed
Note: One channel shown.
     Other channel is identical
Chassis
ground
DUT
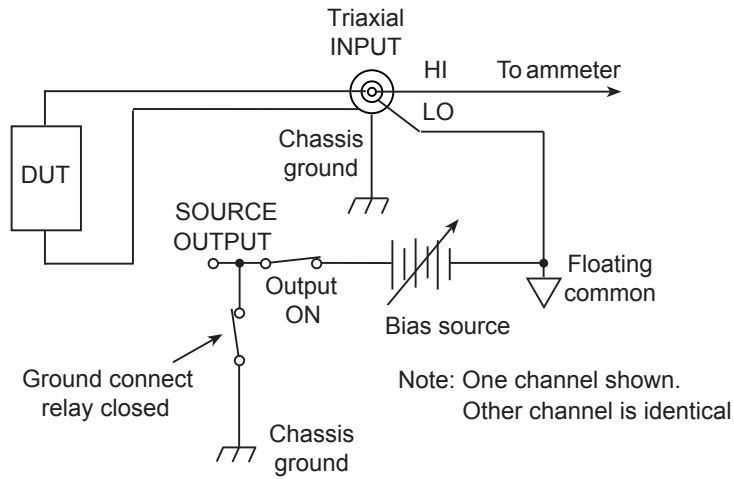
Figure 3-4
Ground connect disabled

Triaxial
INPUT
HI   To ammeter
LO
Chassis
ground
SOURCE
OUTPUT
Output
ON
Bias source
Floating
common
Ground
connect
relay
open
Note: One channel shown.
     Other channel is identical
Chassis
ground
DUT
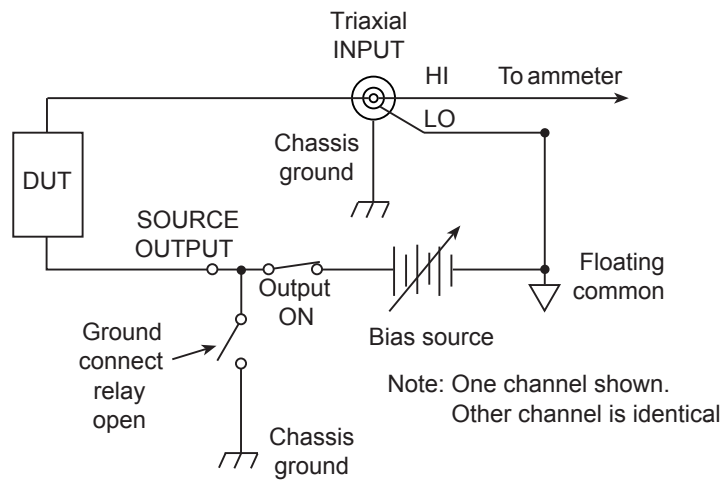
Basic measurement procedure

The following procedures outline measurements on a generic device under test (DUT).

Output control

Use the ON/OFF OUTPUT key to turn both Model 6482 outputs on or off simultaneously for basic source-measure operation. With either channel 1 or channel 2 output on, the blue ON/OFF OUTPUT indicator light will be on. The indicator light turns off when the outputs are turned off.

WARNING   *To prevent electric shock, do not make or break connections to the Model 6482 while it is on.*

Basic measurement circuit configuration

The circuit configuration for the basic measurement procedures that follow is shown in Figure 3-5. In this example, channel 1 connections are shown, but channel 2 connections are similar. See Section 2, "Connections," for detailed connection information.

Figure 3-5
Circuit configuration for basic measurements
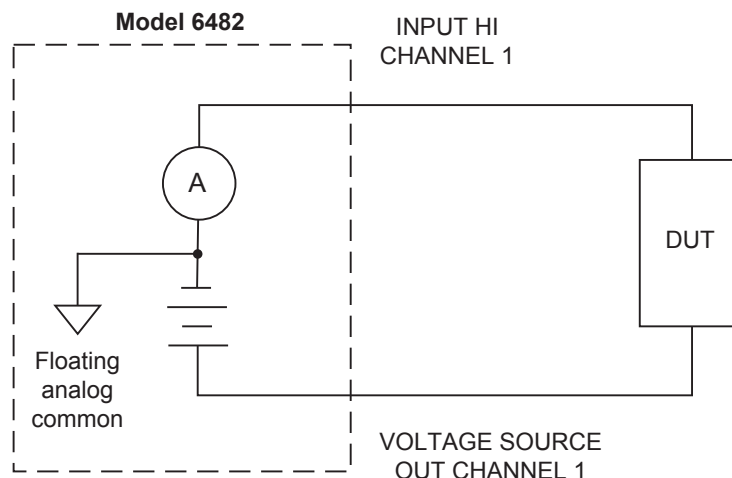
Front-panel measurement procedure

Step 1. Select measurement channel and range.

Select the measurement channel by pressing the **MSR1** (channel 1) or the **MSR2** (channel 2) key. With autoranging turned off, use the RANGE ▲ and ▼ keys to manually select the current measurement range for that channel. You can also press the **AUTO** key to select autoranging.

Step 2. Select source channel and set source level.

The source level is the voltage setting of the selected source (channel 1 or channel 2).

NOTE   *The Model 6482 must be in the edit mode (EDIT indicator ON) to set source values. The edit mode is selected by pressing the **SRC1** or **SRC2** key. The flashing digit for the source value in the Src1 or Src2 display field indicates that the Model 6482 is in the edit mode for that channel. If no editing operation is performed within six seconds, the edit mode times out and is cancelled.*

The EDIT ▲ , ▼ , ◄ , and ▶ keys also enable the edit mode. They choose the last edited source field channel.

When editing the source value, the source is updated immediately, allowing you to adjust the source value while the output is on.

The source value cannot be changed while the Model 6482 *is performing a sweep, which occurs with the output ON and after the SWEEP key is pressed.*

Perform the following steps to edit source values:

1. Press **SRC1** or **SRC2** to enter the edit mode for the specified channel. The flashing digit indicates which value (Src1 or Src2) is presently selected for editing.
2. Use the RANGE ▲ and ▼ keys to select either the 10 V or 30 V range that will accommodate the value you want to set. For best accuracy, use the lowest possible source range.
3. Enter the source value of your choice. There are two methods to edit the value: Value adjust and numeric entry.

NOTE   To clear the source value to 0 V, press the 0000 (MENU) key while in the edit source field.

   • **Value adjust**: To adjust the value, use the EDIT ◄ and ▶ keys to place the cursor at the appropriate position, and use the EDIT ▲ and ▼ keys to increment or decrement the value.
   • **Numeric entry**: When the edit mode is entered, the cursor is located on the most significant digit of the value. From this position, you can key in the value using the number keys (0 through 9). After each number is keyed in, the cursor moves one position to the right. You can use the EDIT ◄ and ▶ keys to place the cursor on a digit to be changed, and press the appropriate number key. The cursor does not have to be on the polarity sign of the value to change polarity. If the 0000 (MENU) key is pressed, the source value will be clear to 0 V.

4. To edit the source on the other channel, press the **SRC1** or **SRC2** key to select it, and then repeat steps 2 and 3.
5. When finished editing source values, press the **ENTER** key or wait six seconds to exit from the edit mode.

Step 3. Turn source outputs on.

Turn the voltage source outputs on by pressing the ON/OFF OUTPUT key. The blue OUTPUT indicator will turn on to indicate that both channel outputs are on.

Step 4. Observe readings on the display.

Observe the current readings in the channel 1 (top left) or channel 2 (top right) display field as appropriate. If in the MSR1 or MSR2 display mode, press the **DISPLAY TOGGLE** key to toggle between channels, or press the **CHANNEL SELECT** key to toggle the displayed channel.

Step 5. Turn source output off.

When finished, turn both voltage source outputs off by pressing the ON/OFF OUTPUT key. The blue OUTPUT indicator light will turn off.

Remote command measurement procedure

Basic measurement procedures can also be performed over a remote interface by sending appropriate commands in the right sequence. The following paragraphs summarize the basic commands and give a simple programming example.

Basic measurement and voltage source commands

Table 3-6 summarizes basic measurement and voltage source commands. See Section 16 for more information on using these commands and many other commands that control these functions.

Table 3-6
Basic measurement and voltage source commands

Command	Description
`:SENSe[1]:CURRent:RANGe <n>`	Set channel 1 current measure range (n = range).
`:SENSe[1]:CURRent:RANGe:AUTO <state>`	Enable or disable channel 1 auto range (state = ON or OFF).
`:SENSe2:CURRent:RANGe <n>`	Set channel 2 measure range (n = range).
`:SENSe2:CURRent:RANGe:AUTO <state>`	Enable or disable channel 2 auto range (state = ON or OFF).
`:SOURce[1]:VOLTage:MODE FIXed`	Select fixed sourcing mode for channel 1 source.
`:SOURce[1]:VOLTage:RANGe <n>`	Select channel 1 source range (n = range, 10 or 30).
`:SOURce[1]:VOLTage <n>`	Set channel 1 source amplitude (n = volts).
`:SOURce2:VOLTage:MODE FIXed`	Select fixed sourcing mode for channel 2 source.
`:SOURce2:VOLTage:RANGe <n>`	Select channel 2 source range (n = range, 10 or 30).
`:SOURce2:VOLTage <n>`	Set channel 2 source amplitude (n = volts).
`:OUTPut[1] <state>`	Turn channel 1 output on or off (state = ON or OFF).
`:OUTPut2 <state>`	Turn channel 2 output on or off (state = ON or OFF).
`:FORMat:ELEMents <name>`	Select reading channel. <name> =CURRent[1] (channel 1), CURRent2 (channel 2).
`:READ?`	Trigger and acquire reading.

Measurement programming example

Table 3-7 summarizes the command sequence for a basic measurement procedure using channel 2. Note that the steps correspond to those listed previously in "Front-panel measurement procedure." These commands set up the Model 6482 as follows:

- Channel 2 measurement range: 2 μA
- Channel 2 source range: 10 V
- Channel 2 source output level: 10 V

NOTE   See Appendix F for a complete program listing.

Table 3-7
Basic measurement command sequence

Step[1]	Action	Commands[2,3]	Comments
		`*RST`	Restore GPIB defaults.
1	Select channel 2 measure range.	`:SENS2:CURR:RANG 2e-6`	Select 2 μA range.
	Select channel 2 current reading.	`:FORM:ELEM CURR2`	Return channel 2 reading.
2	Set channel 2 source range.	`:SOUR2:VOLT:RANG 10`	Select 10 V source range.
	Set channel 2 source amplitude.	`:SOUR2:VOLT 10`	Source 2 output = 10 V.
3	Turn on channel 2 output.	`:OUTP2 ON`	Output on before measuring.
4	Read data.	`:READ?`	Trigger, acquire reading.
5	Turn off channel 2 output.	`:OUTP2 OFF`	Output off after measuring.

[1] Steps correspond to front-panel steps as described in "Front-panel measurement procedure."
[2] Commands must be sent in order given.
[3] Instrument must be addressed to talk after :READ? to acquire data.

Using the analog outputs

Each analog output provides a -10 V to +10 V DC output analogous to the input current. The analog output signal will nominally source a ±10 V proportional, but inverted, signal to the input current on all measurement ranges. For example, with a +1 mA current on the 2 mA measurement range, the analog output voltage would be -5 V. Table 3-8 list typical analog output voltages for various ranges and input currents.

Table 3-8
Analog output voltage examples

Range	Input current	Analog output
20 nA	+10 nA	-5 V
2 µA	-2 µA	+10 V
20 µA	+7 µA	-3.5 V
2 mA	+0.5 mA	-2.5 V
20 mA	-15 mA	+7.5 V

This page left blank intentionally.

Range, Digits, Speed, and Filters

In this section:

Range and digits

This topic discusses available ranges, maximum readings, ranging limitations, manual and autoranging, and display resolution.

Measurement range

The selected measurement range affects the accuracy of the measurements and the maximum signal that can be measured.

Available ranges

Table 4-1 lists the available current measurement ranges, resolution values, and maximum readings for each Model 6482 channel.

NOTE   *The current measurement range can be set individually for each channel.*

Table 4-1
Current measurement ranges

Model 6482 I-range	Maximum resolution	Maximum reading
2 nA	1 fA	±2.1 nA
20 nA	10 fA	±21 nA
200 nA	100 fA	±210 nA
2 µA	1 pA	±2.1 µA
20 µA	10 pA	±21 µA
200 µA	100 pA	±210 µA
2 mA	1 nA	±2.1 mA
20 mA	10 nA	±21 mA

Maximum readings

As shown in Table 4-1, the full scale input for each current measurement range is 105 percent of the selected range. For example, ±21 mA is the full scale reading for the 20 mA range. Input levels that exceed the maximum levels cause the Oflo message to be displayed.

Manual ranging

Press the **MSR1** or **MSR2** key, and then use the RANGE ▲ and ▼ keys to select a fixed channel 1 or channel 2 manual range.

NOTE   *Use the lowest range possible without causing an overflow to ensure best accuracy and resolution.*

Autoranging

Press the **MSR1** or **MSR2** key, and then press the **AUTO** RANGE key to enable autoranging on channel 1 or channel 2 respectively. The AUTO indicator turns on when autoranging is selected. With autoranging selected, the instrument automatically chooses the best range to measure the applied signal. In 2-channel, RATIO, or DELTA display modes, the AUTO indicator appears if either channel is in autorange. Pressing the **AUTO** RANGE key in the 2-channel display mode toggles autorange ON or OFF on both channels.

NOTE   *With the median filter enabled, autoranging can be very slow. See "Median filter," page Section 4-7 for details.*

Autorange limits

Upper and lower autorange limits are included to support the autorange change mode. The upper limit must be greater than or equal to the lower limit, and the lower limit must be less than or equal to the upper limit. If the lower limit is equal to the upper limit, autoranging is effectively disabled. When autoranging is disabled, you can manually change to any range below the lower limit or any range above the upper limit.

Setting autorange limits

To set the upper or lower autorange limit:

1. Press the **CONFIG** key.
2. To set the upper limit, press RANGE ▲ key to select the ULIMIT prompt.
   To set the lower limit, press RANGE ▼ key to select the LLIMIT prompt.
3. At the prompt, use the RANGE ▲ or ▼ keys again to set the limit.

Autorange speed limitations

With autorange enabled, a minimum of two extra readings are required to move up range, and one additional reading is needed to move down range. No additional readings are required if no range change is required.

Autorange operation with range limits

The autorange limits are only evaluated when an autorange condition exists and forces the limits to be re-evaluated. Only the upper limit is evaluated if the instrument needs to up-range; similarly, only the lower limit is evaluated if the instrument needs to down-range. Both limits are not checked simultaneously for any given autorange evaluation.

For example, if the instrument is on a higher manual range that is above the programmed upper range limit and autorange is then enabled, the instrument will not re-evaluate the range limits until a change in the input signal level forces a range change. In this example, the instrument would stay on a range above the programmed upper limit with autorange enabled until the input signal level changes accordingly.

Digits

The display resolution of the measured reading depends on the DIGITS setting. This setting is global, which means the digits setting selects display resolution for both channels.

The DIGITS setting has no effect on the remote reading format. The number of displayed digits does not affect accuracy or speed. Those parameters are controlled by the SPEED setting.

Setting display resolution

There are two ways to set display resolution:

- **DIGITS**: Press the **DIGITS** key until the required resolution is displayed.
- **CONFIG DIGITS**: Press the **CONFIG** key and then the **DIGITS** key to display the digits menu. Place the cursor to select the resolution (3.5, 4.5, 5.5, or 6.5) and press the **ENTER** key.

NOTE   *The DIGITS setting affects only the single-channel display mode; it does not affect dual-channel display. Changing SPEED changes DIGITS, but changing DIGITS does not change SPEED.*

Remote range and digits programming

Table 4-2 summarizes the commands necessary to control range and digits. See Section 16 for more details on these commands.

Table 4-2
Range and digits commands

Commands	Description
`:SENSe[1]:CURRent:RANGe <n>`	Select channel 1 manual amps range (n = range).
`:SENSe[1]:CURRent:RANGe:AUTO <state>`	Enable or disable Channel 1 autorange (state = ON or OFF).
`:SENSe[1]:CURRent:RANGe:AUTO:ULIMit <n>`	Set upper limit for Channel 1 autorange (n = range).
`:SENSe[1]:CURRent:RANGe:AUTO:LLIMit <n>`	Set lower limit for Channel 1 autorange (n = range).
`:SENSe2:CURRent:RANGe <n>`	Select channel 2 manual amps range (n = range).
`:SENSe2:CURRent:RANGe:AUTO <state>`	Enable or disable Channel 2 autorange (state = ON or OFF).
`:SENSe2:CURRent:RANGe:AUTO:ULIMit <n>`	Set upper limit for Channel 2 autorange (n = range).
`:SENSe2:CURRent:RANGe:AUTO:LLIMit <n>`	Set lower limit for Channel 2 autorange (n = range).
`:DISPlay:DIGits <n>`	Set display digits (n = 4, 5, 6, or 7).

Range and digits programming example

Table 4-3 shows a programming example for controlling range and digits. The Model 6482 is set up as follows:

- Channel 1 autorange: ON
- Channel 1 upper range limit: 2 mA
- Channel 1 lower range limit: 200 nA
- Display digits: 5½

Table 4-3
Range and digits programming example

Command	Description
`*RST`	Restore GPIB defaults.
`:SOUR1:VOLT 10`	Source 10 V on Channel 1.
`:SENS1:CURR:RANG:AUTO ON`	Enable Channel 1 autorange.
`:SENS1:CURR:RANG:AUTO:ULIM 2e-3`	2 mA upper range limit.
`:SENS1:CURR:RANG:AUTO:LLIM 200e-9`	200 nA lower range limit.
`:DISP:DIG 5`	5½ display digits.
`:OUTP1 ON`	Turn on output.
`:READ?`	Trigger and acquire reading.
`:OUTP1 OFF`	Turn off output.

Speed

The Speed/Accuracy menu is used to set the integration time of the analog-to-digital converter (period of time the input signal is measured). The integration time affects the usable digits, the amount of reading noise, and the reading rate of the instrument. The integration time is specified in parameters based on the number of power line cycles (NPLC), where 1 PLC for 60 Hz is 16.67 ms (1/60) and 1 PLC for 50 Hz is 20 ms (1/50).

In general, the fastest integration time (FAST; 0.01 PLC) results in increased reading noise and fewer usable digits. The slowest integration time (HI ACCURACY; 10 PLC) provides the best common-mode and normal-mode noise rejection. In between settings are a compromise between speed and noise. The default power-on speed setting is NORMAL (1 PLC).

Setting speed

Speed is set from the SPEED ACCURACY MENU and is structured as follows. Use Section 1, "Rules to navigate menus," to check and/or change the speed setting.

SPEED-ACCURACY MENU

Press the **SPEED** key (or press the **CONFIG** key and then the **SPEED** key) to display the menu (Figure 4-1).

- **FAST**: Sets speed to 0.01 PLC and sets display resolution to 3½ digits.
- **MED**: Sets speed to 0.10 PLC and sets display resolution to 4½ digits.
- **NORMAL**: Sets speed to 1.00 PLC and sets display resolution to 5½ digits.
- **HI ACCURACY**: Sets speed to 10.00 PLC and sets display resolution to 6½ digits.
- **OTHER**: Use to set speed to any PLC value from 0.01 to 10. Display resolution is not changed when speed is set with this option.

NOTE   The SPEED setting is global and affects both channels. After setting speed, display resolution for the single-channel display mode can be changed using the DIGITS key.

Figure 4-1
Speed configuration menu tree
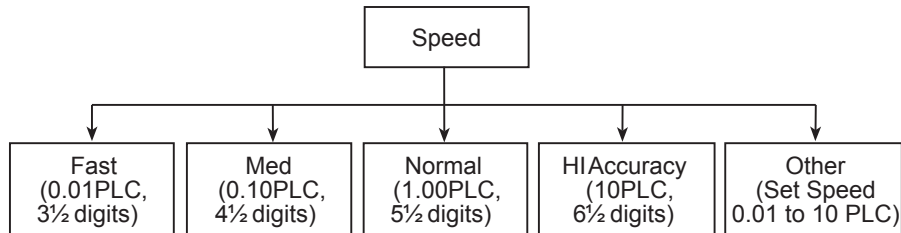
Remote speed programming

Speed commands

Table 4-4 summarizes commands to control speed. Although commands for both channel 1 and channel 2 are included, the NPLC setting is global and affects both channels. See Section 16 for more information.

Table 4-4
Speed commands

Command	Description
`:SENSe[1]:CURRent:NPLCycles <n>`	Set speed (n = PLC, 0.01 to 10).
`:SENSe2:CURRent:NPLCycles <n>`	Set speed (n = PLC, 0.01 to 10).

Speed programming example

Send the following command to set the speed for both channel 1 and channel 2 to 10 PLC:

```
:SENS1:CURR:NPLC 10
```

Filters

Filtering stabilizes noisy measurements caused by noisy input signals. However, the more filtering that is used, the slower the measurement process becomes. The Model 6482 uses two stages of filtering: Average and median. The displayed, stored, or transmitted reading is the result of the filtering processes.

Filter stages

The Model 6482 uses a two-stage filtering system (see Figure 4-2). The first stage applies the average filter (moving or repeat) to the measurement conversions. The second stage applies the median filter to the output of the first stage. When a filter stage is disabled, a reading passes through it.

Figure 4-2
Two-stage filtering

*Repeat or moving
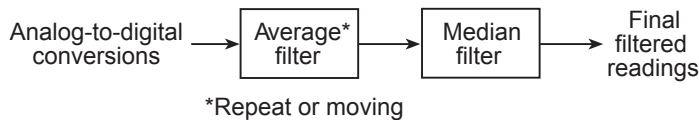
Each filter stage uses a stack to temporarily store readings to be filtered. The size of a stack, which is set by the user, determines how many readings will be filtered. A stack size of one (1) disables that filter.

NOTE   *A source level change due to a sweep step resets the filters. That is, readings are flushed from stack, and the filtering process starts over at the beginning. When not sweeping, source level changes do not reset the filters.*

Any range change will also reset the filters.

Repeat filter

The repeat filter places the specified number of measurement conversions into a stack and averages them to yield a single repeat filter reading. The stack is then cleared, and the process starts over. For example, if the repeat count (stack size) is 10, every 10 measurement conversions will yield a single reading. Figure 4-3 illustrates the repeat filter process. The maximum count (stack size) for the repeat filter is 100. Note that setting the count to one (1) disables the repeat filter.

Choose the repeat filter for sweeping so that readings for other source levels are not averaged with the present source level.

Figure 4-3
Repeat filter (count 10)
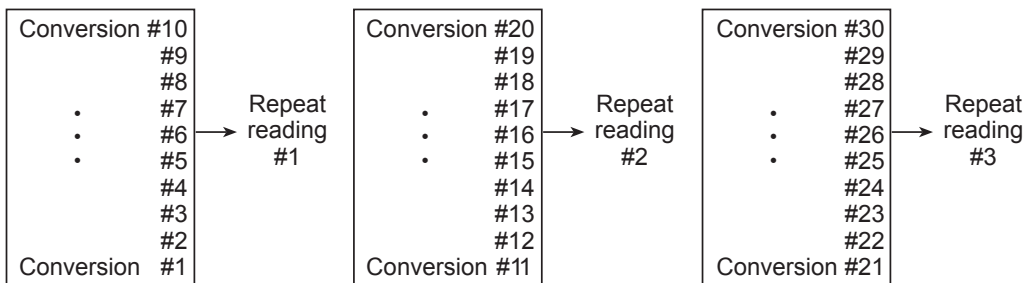
Median filter

The median filter is used to pass the median, or "middle-most," reading from a group of readings that are arranged according to size. For example, assume the following readings:

2 mA, 1 nA, 3 nA

The readings are placed in a stack, rearranged in ascending order as follows:

1 nA, 3 nA, 2 mA

From the above readings, it can be seen that 3 nA is the median reading. Therefore, the 3 nA reading is allowed to pass and the other two readings are discarded (filtered out). The median filter provides a good method to reject noise.

The number of reading samples (stack size) for the median filter is determined by the selected rank (0 to 5), as follows:

Sample readings = 2n + 1

Where: n is the selected rank (0 to 5)

From the above equation, it can be seen that the minimum number of sample readings is 1 (n = 0) and the maximum number is 11 (n = 5). The following table shows the number of sample readings for each rank setting.

Rank setting	Number of sample readings
0	1
1	3
2	5
3	7
4	9
5	11

The first-in, first-out stack for the median filter operates as a moving type after it fills. For example, if the median filter is configured to sample 11 readings (rank setting 5), as shown in Figure 4-4, the first filtered reading will be calculated (and displayed) after 11 readings are acquired and placed in the filter stack. Each subsequent reading will then be added to the stack (oldest reading discarded) and another median filter reading will be calculated and displayed.

NOTE *With autorange enabled, a range change cannot occur until a reading is yielded by the median filter process. Therefore, autoranging could be very slow when the median filter is enabled.*

If the repeat filter is enabled, the median filter operation will not start until after the repeat filter operation yields a reading. In other words, after a repeat filter reading is yielded, that reading will then be sent to the median filter stack.

Figure 4-4
Median filter (rank 5)
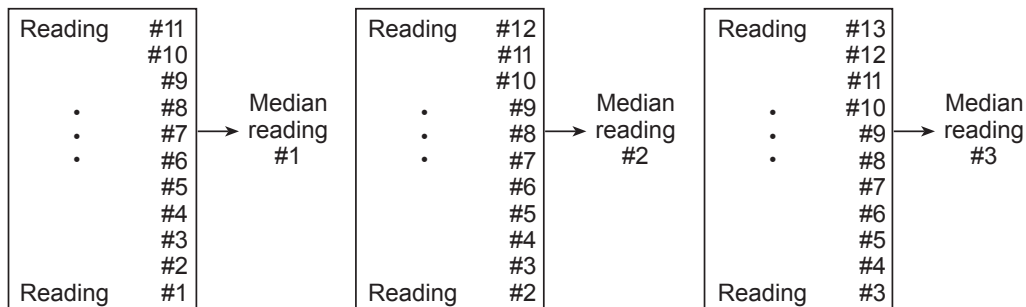
Moving filter

The moving average filter uses a first-in, first-out stack. When the stack (filter count) becomes full, the readings are averaged, yielding a filtered reading. For each subsequent reading placed into the stack, the oldest reading is discarded. The stack is reaveraged, yielding a new reading.

When the filter is first enabled, the stack is empty. Note that a moving filter reading is not yielded until the stack is full. The first reading is placed in the stack and is then copied to fill the other stack locations. Therefore, the first filtered reading is the same as the first reading that entered the stack. Now the normal moving average filter process can continue. Note that a true average is not yielded until the stack is filled with new readings (no copies in stack). For example, in Figure 4-5, it takes ten filtered readings to fill the stack with new readings. The first nine filtered readings are calculated using copied readings.

Figure 4-5
Moving filter (count 10)
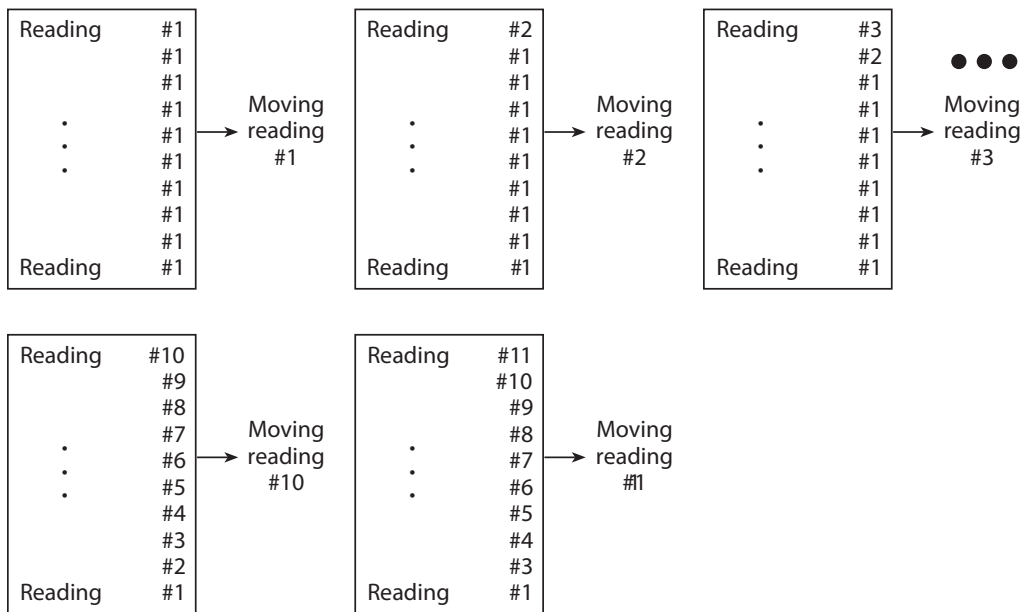
Advanced filter: The advanced filter is part of the moving filter. With the advanced filter enabled, a user-programmable noise "window" is used with the moving filter. The noise window, which is expressed as a percentage of range (0 to 105 percent), allows a faster response time to large signal step changes. As previously explained, if the readings are within the noise window, the moving filter operates normally. If, however, a reading falls outside of the window, the stack is flushed of old readings and filled with the new reading.

For example, assume the window is set to 10 percent and the 20 mA range is selected. Therefore, the noise window is ±2 mA (20 mA × 10% = 2 mA). Also assume the first reading is 2 mA. During normal filter operation, the stack is filled with that reading. If each subsequent reading is within ±2 mA of the previous reading, the filter operates normally. Now assume a 10 mA noise spike occurs. This noise window violation causes the stack to flush out the old readings and fill it with 10 mA reading.

NOTE   *If the repeat or median filter is enabled, the moving filter operation will not start until after the previous filter yields a reading. In other words, after a reading is yielded from the repeat or median filter, that reading will then be sent to the moving filter stack.*

Filter configuration

Using Table 4-5 and Figure 4-6 as a guide, configure the filter as follows:

NOTE   *The filter setting is global and affects both channels.*

1. Press the **CONFIG** key and then the **FILTER** key to access the filter configuration menu.
2. Select **AVERAGE-MODE**, and then choose the REPEAT or MOVING filter.
3. Place the cursor on **AVERAGE-COUNT** and press the **ENTER** key to display the present average count for the filter.
4. Use the ◂ , ▸ , ▴ , and ▾ keys to display the count (1 to 100), and then press the **ENTER** key.
5. Place the cursor on **ADVANCED** and press the **ENTER** key. The blinking cursor will indicate the state of advanced filter.
6. Use the ◂ or ▸ key to place the cursor on the advanced filter selection (DISABLE or ENABLE), and then press the **ENTER** key.
7. If you enabled the advanced filter, use the ◂ , ▸ , ▴ , and ▾ keys to select and display the noise window (**0 %** to **105 %**), and then press the **ENTER** key.
8. Place the cursor on **MEDIAN RANK** and press the **ENTER** key to display the present median rank for the median filter.
9. Place the cursor on the rank value (0 to 5) and press the **ENTER** key. Note that a rank of 0 disables the median filter.
10. Press the **EXIT** key to return to the previous menu.

Filter control

When filtering is applied to the input signal, the FILT indicator will be on. The FILTER key is used to control filtering. Pressing the **FILTER** key turns on the FILT indicator to indicate that the filter configuration is being applied to the input. Pressing the **FILTER** key a second time turns the FILT indicator off to indicate that filtering is turned off.

Table 4-5
Filter configuration menu

Menu item	Description
CONFIG > **FILTER**	Configure filter.
CONFIGURE FILTERING	
AVERAGE-MODE	Select average mode filter.
MOVING	Moving average.
REPEAT	Repeat average.
AVERAGE-COUNT	Set average filter count (1 to 100).
ADVANCED	Enable or disable advanced filter.
DISABLE	Disable advanced filter.
ENABLE	Enable advance filter.
MEDIAN-RANK	Select median rank (0 to 5).

Figure 4-6
Configure filtering menu tree
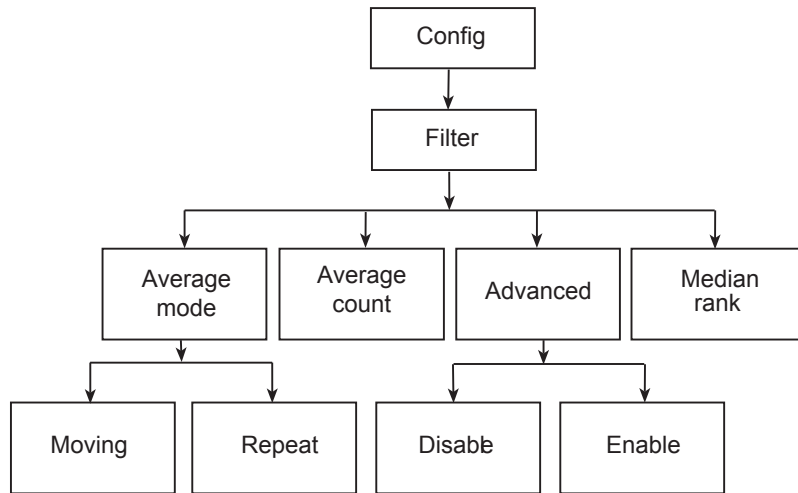
Remote filter programming

Filter commands

Table 4-6 summarizes filter commands. See Section 16 for more details.

Table 4-6
Filter commands

Commands*	Description
Average filter:	
[:SENSe[1]]:AVERage:COUNt <n>	Set Channel 1 average filter count (n = count, 1 to 100).
[:SENSe[1]]:AVERage:TCONtrol <name>	Select Channel 1 filter type (name = MOVing or REPeat).
[:SENSe[1]]:AVERage[:STATe] <state>	Enable or disable Channel 1 filter (state = ON or OFF).
[:SENSe[1]]:AVERage:ADVanced:NTOLerance <NRf>	Set Channel 1 advanced filter noise window in percentage (NRf = noise window, 0 to 105).
[:SENSe[1]]:AVERage:ADVanced[:STATe] <state>	Enable or disable Channel 1 advanced filter (state = ON or OFF).
:SENSe2:AVERage:COUNt <n>	Set Channel 2 average filter count (n = count, 1 to 100).
:SENSe2:AVERage:TCONtrol <name>	Select Channel 2 filter type (name = MOVing or REPeat).
:SENSe2:AVERage[:STATe] <state>	Enable or disable Channel 2 filter (state = ON or OFF).
:SENSe2:AVERage:ADVanced:NTOLerance <NRf>	Set Channel 2 advanced filter noise window in percentage (NRf = noise window, 0 to 105).
:SENSe2:AVERage:ADVanced[:STATe] <state>	Enable or disable Channel 2 advanced filter (state = ON or OFF).
Median filter:	
[:SENSe[1]]:MEDian:RANK <NRf>	Set Channel 1 median filter rank (NRf = rank, 0 to 5).
[:SENSe[1]]:MEDian[:STATe] <state>	Enable or disable Channel 1 median filter (state = ON or OFF).
:SENSe2:MEDian:RANK <NRf>	Set Channel 2 median filter rank (NRf = rank, 0 to 5).
:SENSe2:MEDian[:STATe] <state>	Enable or disable Channel 2 median filter (state = ON or OFF).

* Because filter commands are global, :SENSe[1] (channel 1) commands also affect equivalent :SENSe2 (channel 2) settings.

Filter programming example

Table 4-7 summarizes the command sequence to program channel 1 filter attributes as follows:

- Average filter: Off
- Median filter state: On
- Median filter rank: 5
- Average filter state: On
- Average filter type: Moving
- Average filter count: 20
- Advanced filter state: On
- Advance filter tolerance: 10 percent

Table 4-7
Filter programming example

Command	Description
:MED:RANK 5	Set median rank to 5.
:MED ON	Enable median filter.
:AVER:TCON MOV	Select moving filter type.
:AVER:COUN 20	Set moving count to 20.
:AVER ON	Enable average filter.
:AVER:ADV OFF	Disable advanced filter.
:AVER:ADV:NTOL 10	Set filter tolerance to 10%.

Relative, Math, Ratio, and Delta

In this section:

Relative

The relative offset (REL) feature can be used to set offsets to zero (0) or subtract a baseline reading from present and future readings. With relative offset enabled, subsequent readings are the difference between the actual input value and the relative offset value, as follows:

Displayed reading = Actual input - Relative offset value

When relative offset is enabled, the REL indicator turns on.

Once a relative offset value is established for a measurement function, the value is the same for all ranges. For example, if 1 mA is set as a relative offset value on the 20 mA range, the relative offset value is also 1 mA on the 2 mA current range.

Selecting a range that cannot accommodate the relative offset value does not cause an overflow condition, but it also does not increase the maximum allowable input for that range. For example, on 2 mA range, the Model 6482 still overflows for a >2.1 mA input.

NOTE   *In single-channel display mode, the REL indicator turns on for the displayed channel. In dual-channel display mode, the REL indicator turns on if relative offset is enabled for either channel.*

Front panel relative offset

Enabling and disabling relative offset

Relative offset can be used to set offsets to zero (0) or to establish a zero baseline on the selected measurement channel by pressing the **REL** key. The reading (which becomes the relative offset value) is subtracted from itself. As a result, a zero (0) reading is displayed. Pressing the **REL** key a second time disables the relative offset feature.

Defining a relative offset value

A unique relative offset value can be established for the selected channel from the front panel as follows:

1. Press **MSR1** or **MSR2** to select either channel 1 or channel 2 measurement.
2. Press the **CONFIG** key and then the **REL** key. The present relative offset value will be displayed.
3. Set the relative offset value. See Section 1, "Rules to navigate menus," for details.
4. With the relative offset value displayed, press the **ENTER** key. The Model 6482 will return to the normal display with relative offset enabled. The reading will reflect the defined relative offset value.

Using the relative offset feature in dual-channel mode

The REL key does not function while the display is in the dual-channel, RATIO, or DELTA display modes. To view two relative offset readings simultaneously:

1. Press the **MSR1** key.
2. Press the **REL** key.
3. Press the **MSR2** key.
4. Press the **REL** key.
5. Press the **TOGGLE** key to toggle the display between the two channels.

Remote relative offset programming

Relative offset commands

Table 5-1 summarizes relative offset commands. See Section 16 for additional information.

Table 5-1
Relative offset commands

Command	Description

`:CALCulate3:NULL:OFFSet <n>`
`:CALCulate3:NULL:STATe <state>`
`:CALCulate3:NULL:ACQuire`
`:CALCulate3:FEED <name>`
`:CALCulate4:NULL:OFFSet <n>`
`:CALCulate4:NULL:STATe <state>`
`:CALCulate4:NULL:ACQuire`
`:CALCulate4:FEED <name>`
	Define channel 1 null relative offset value (n = value).
	Enable or disable channel 1 relative offset (state = ON or OFF).
	Acquire channel 1 relative offset value.
	Select input feed (name = SENS1 or CALC1).
	Define channel 2 null relative offset value (n = value).
	Enable or disable channel 2 relative offset (state = ON or OFF).
	Acquire channel 2 relative offset value.
	Select input feed (name = SENS2 or CALC2).

Relative offset programming example

Table 5-2 lists commands for setting up and enabling the relative offset feature. These commands set up the Model 6482 as follows:

- Channel 1 relative offset value: 1e-3
- Channel 1 feed: SENS1 (current function block)
- Channel 1 relative offset state: Enabled

Table 5-2
Relative offset programming example

Command	Description

`:CALC3:NULL:OFFS 1e-3`
`:CALC3:FEED SENS1`
`:CALC3:NULL:STAT ON`
`:OUTP1 ON`
`:INIT`
`:CALC3:DATA?`
`:OUTP1 OFF`
	Relative offset value = 1e-3.
	Current measure feed.
	Enable relative offset.
	Output on.
	Trigger reading.
	Get reading.
	Turn off output.

Measurement math functions

This section provides detailed information on the following math functions: I/V, V/I, MX + B, and electrical power (V x I).

Math functions

Each Model 6482 channel has built-in math functions to calculate the following:

- I/V
- V/I
- MX + B
- Electrical power

NOTE   *The math functions use the current and voltage from the same channel for calculations.*

I/V

This math function computes the ratio between the selected channel current measurement and the voltage bias source value:

Reading = I/V

Where: *I*   = Channel 1 or channel 2 current
        V   = Channel 1 or channel 2 voltage bias

V/I

This math function computes the ratio between the selected channel voltage bias source value and the current measurement:

Reading = V/I

Where: *V*   = Channel 1 or channel 2 voltage bias
        I   = Channel 1 or channel 2 current

MX + B

This math function multiplies the measured current by an offset factor and adds an offset value as follows:

Reading = MX + B

Where: M = Gain (slope) factor
X = Channel 1 or channel 2 measured current
B = Offset value

Electrical power

This math function calculates power using the bias voltage and measured current values as follows:

Reading = V $\times$ I

Where: V = Channel 1 or channel 2 voltage
I = Channel 1 or channel 2 current

Front panel math functions

Using Table 5-3 as a guide, configure measurement math functions as follows:

1. Press the **CONFIG** key and then the **MSR1** or **MSR2** key to select channel 1 or channel 2, respectively.
2. Using the EDIT ◀ and ▶ keys, place the cursor on the function, and then press the **ENTER** key.
3. From normal display, press the **MSR1** or **MSR2** key to display the function reading.
4. To cancel the function and return to the normal current measurement display, repeat steps 1 and 2, but select I in the configuration menu.

Table 5-3
Measurement function configuration menu

Configuration menu item	Description
CONFIG > **MSR1**	Configure channel 1 measurement.
CONFIG MSR1 BUTTON	
I	Current measurement.
I/V	Conductance (I/V) measurement (channel 1I / channel 1V).
V/I	Resistance (V/I) measurement (channel 1V / channel 1I).
MX + B_UNIT	MX + B measurement.
P ⚡	Electrical power (V $\times$ I) measurement (channel 1V $\times$ channel 1I).
CONFIG > **MSR2**	Configure channel 2 measurement.
CONFIG MSR2 BUTTON	
I	Current measurement.
I/V	Conductance (I/V) measurement (channel 2I / channel 2V).
V/I	Resistance (V/I) measurement (channel 2V / channel 2I).
MX + B_UNIT	MX + B measurement.
P ⚡	Electrical power (V $\times$ I) measurement (channel 2V $\times$ channel 2I).

Remote math functions

Table 5-4 summarizes commands to control the measurement math functions through a remote interface. See the :CALC1 and :CALC2 subsystems in Section 16 for detailed information.

Table 5-4
Math function commands

Command	Description
`:CALCulate[1]:DATA?` `:CALCulate[1]:FORMat <name>`	Request channel 1 math reading. Set Channel 1 math function. Name = MXB[1] (MX + B), COND[1] (I1/V1), POWER[1]. (I1 * V1), or RES[1] (V1/I1).
`:CALCulate[1]:KMATh:MBFactor <n>`	Channel 1 MX + B offset (n = B).
`:CALCulate[1]:KMATh:MMFactor <n>`	Channel 1 MX + B slope (n = M).
`:CALCulate[1]:KMATh:MUNits "<string>"`	Channel 1 MX + B units. String = Units.
`:CALCulate[1]:STATe <state>`	Enable or disable channel 1 math. State = ON or OFF.
`:CALCulate2:DATA?`	Request channel 2 math reading.
`:CALCulate2:FORMat <name>`	Set Channel 2 math function. Name = MXB2 (MX + B), COND2 (I2/V2), or POWER2 (I2 * V2), RES2 (V2/I2).
`:CALCulate2:KMATh:MBFactor <n>`	Channel 2 MX + B offset (n = B).
`:CALCulate2:KMATh:MMFactor <n>`	Channel 2 MX + B slope (n = M).
`:CALCulate2:KMATh:MUNits "<string>"`	Channel 2 MX + B units. String = units.
`:CALCulate2:STATe <state>`	Enable or disable channel 2 math. State = ON or OFF.
`:FORMat:ELEMents:CALCulate <name>`	Define CALC data elements. Name = CALCulate, TIME, or STATus.
`:INIT`	Trigger CALC readings (must use before sending CALCn:DATA?).

Math function programming example

Table 5-5 summarizes commands that program the following math function parameters:

- Channel 2 math function: MX + B
- Channel 2 MX + B slope (M): 10
- Channel 2 MX + B offset (B): 5

Table 5-5
Math function programming example

Command	Description
`:CALC2:FORM MXB2`	Select Channel 2 MX + B function.
`:CALC2:KMAT:MBF 5`	Channel 2 offset (B) = 5.
`:CALC2:KMAT:MMF 10`	Channel 2 slope (M) = 10.
`:CALC2:STAT ON`	Enable Ch 2. math.
`:OUTP2 ON`	Turn on Channel 2 output.
`:INIT`	Trigger math reading.
`:CALC2:DATA?`	Request Channel 2 MX + B result.
`:OUTP2 OFF`	Output 2 off.

RATIO and DELTA

This topic discusses the ratio and delta functions, which can be used to calculate the ratio or difference between channel 1 and channel 2 measurement function results.

RATIO functions

The Model 6482 has built-in RATIO functions to calculate the following:

- MSR1/MSR2
- MSR2/MSR1

MSR1/MSR2

This function computes the ratio between the channel 1 measurement function (MSR1) and the channel 2 measurement function (MSR2):

RATIO = MSR1/MSR2

Where: *MSR1* = Channel 1 measurement function
       MSR2 = Channel 2 measurement function

MSR2/MSR1

This function computes the ratio between the channel 2 measurement function and the channel 1 measurement function:

RATIO = MSR2/MSR1

where: *MSR1* = Channel 1 measurement function
       MSR2 = Channel 2 measurement function

DELTA functions

The Model 6482 has built-in DELTA functions to calculate the following:

- MSR1-MSR2
- MSR2-MSR1

MSR1-MSR2

This function computes the difference between the channel 1 measurement function and the channel 2 measurement function:

DELTA = MSR1-MSR2

Where: *MSR1* = Channel 1 measurement function
       MSR2 = Channel 2 measurement function

MSR2-MSR1

This function computes the difference between the channel 2 measurement function and the channel 1 measurement function:

DELTA = MSR2-MSR1

Where: *MSR1* = Channel 1 measurement function
       MSR2 = Channel 2 measurement function

Front panel RATIO and DELTA configuration

Using Table 5-6 as a guide, configure RATIO and DELTA functions as follows:

1. Press the **CONFIG** key then the **RATIO** or **DELTA** key to select the function to be configured.
2. Using the EDIT ◂ and ▸ keys, place the cursor on the function, and then press the **ENTER** key.
3. From normal display, press the **RATIO** or **DELTA** key to display the function reading.
4. To cancel the function and return to normal display, press the **MSR1** or **MSR2** key.

Table 5-6
RATIO and DELTA configuration menus

Configuration menu item	Description
CONFIG RATIO	Configure RATIO function.
CONFIG RATIO BUTTON	
MSR1/MSR2	MSR1/MSR2 function.
MSR2/MSR1	MSR2/MSR1 function.
CONFIG DELTA	Configure DELTA function.
CONFIG DELTA BUTTON	
MSR1-MSR2	MSR1-MSR2 function.
MSR2-MSR1	MSR2-MSR1 function.

Remote RATIO and DELTA

Table 5-7 summarizes commands to control the RATIO and DELTA function through a remote interface. See the :CALC5 and :CALC6 subsystems in Section 16 for detailed information.

Table 5-7
RATIO and DELTA function commands

Command	Description*
:CALCulate5:DATA?	Request ratio reading.
:CALCulate5:FORMat <name>	Set ratio function.
	Name =C3C4 (CALC3/CALC4)
	C4C3 (CALC4/CALC3)
:CALCulate5:STATe <state>	Enable or disable ratio. State = ON or OFF.
:CALCulate6:DATA?	Request delta reading.
:CALCulate6:FORMat <name>	Set delta function.
	Name =C3C4 (CALC3-CALC4)
	C4C3 (CALC4-CALC3)
:CALCulate6:STATe <state>	Enable or disable delta. State = ON or OFF.
:FORMat:ELEMents:CALCulate <name>	Define CALC data elements. Name = CALCulate, TIME, or STATus.
:INIT	Trigger CALC readings (must use before sending CALCn:DATA?).

*CALC1 (Channel 1 math) feeds into CALC3 (Channel 1 relative offset). CALC2 (Channel 2 math) feeds into CALC4 (Channel 2 relative offset). See CALCulate subsystems in Section 17 for details.

RATIO and DELTA function programming example

Table 5-8 summarizes commands that program the following RATIO and DELTA functions:

- RATIO function: CALC3/CALC4
- DELTA function: CALC4-CALC3

Table 5-8
RATIO and DELTA programming example

Command	Description
:CALC5:FORM C3C4	Select CALC3/CALC4 ratio.
:CALC5:STAT ON	Enable ratio function.
:OUTP1 ON;OUTP2 ON	Turn on outputs.
:INIT	Trigger ratio reading.
:CALC5:DATA?	Request ratio reading.
:CALC6:FORM C4C3	Select CALC4-CALC3 delta.
:CALC6:STAT ON	Enable delta function.
:INIT	Trigger delta reading.
:CALC6:DATA?	Request delta reading.
:OUTP1 OFF;OUTP2 OFF	Turn off outputs.

This page left blank intentionally.

In this section:

Data store overview

The Model 6482 has a data store (buffer) to store from 1 to 3000 voltage bias source values and measurement readings for both channels. Each reading includes the buffer location number and a timestamp.

The data store buffer also provides statistical data on the measured readings stored in the buffer. These include minimum, maximum, mean, and standard deviation.

NOTE   The buffer automatically stores readings for sweeps having a finite sweep count.

Front-panel data store

This topic describes basic data store (buffer) capabilities as accessed from the front panel.

Storing readings

Perform the following steps to store readings:

1. Select the measurement configuration by pressing the **MSR1**, **MSR2**, **RATIO**, or **DELTA** keys.

NOTE   Data storage cannot be performed when the dual-channel display mode is selected.

2. Set the source values using the **SRC1** and **SRC2** keys, and then turn on the output.
3. Press the **STORE** key.
4. Use the EDIT ▲ , ▼ , ◀ , and ▶ keys to specify the number of readings to store in the buffer.
5. Press the **ENTER** key. The asterisk (*) indicator turns on to indicate that a data storage operation is in process (the indicator will turn off when the storage operation is finished).
6. After the storage operation is finished, press the **ON/OFF OUTPUT** key to turn the output off (the blue OUTPUT indicator will turn off).

Recalling readings

Readings stored in the buffer are displayed by pressing the RECALL key. The source and measure readings are positioned at the left side of the display; the buffer location number and timestamp are positioned at the right side.

NOTE   RECALL cannot be used with the dual-channel display mode. RECALL will display buffer data of the presently selected function (MSR1, MSR2, RATIO, or DELTA). You can display these different functions while the recall mode is active by pressing the appropriate function key.

Buffer location number

The buffer location number indicates the memory location of the source-measure reading. Location #0000 indicates that the displayed source-measure reading is stored at the first memory location. If limit testing was performed, a "P" or an "F" will precede the buffer location number to indicate the pass/fail result of the test. Limit testing is covered in Section 10.

Timestamp

The first source-measure reading stored in the buffer (#0000) is timestamped at 0000000.000 seconds. Subsequent readings can be recalled in absolute or DELTA timestamp format. For the absolute format, the timestamp references readings to zero (0) seconds. For the DELTA format, the timestamp indicates the time between the displayed reading and the reading before it. To set the timestamp format, see "Timestamp format," page Section 6-4.

Displaying other buffer readings

To display the other readings stored in the buffer, display the memory location number. The EDIT ▲ and ▼ keys increment and decrement the selected digit of the location number. Cursor position is controlled by the EDIT ◄ and ► keys. When scrolling forward past the last stored reading, the buffer wraps to the first stored reading. Conversely, when scrolling in reverse past the first stored reading, the buffer wraps to the last stored reading.

The memory location number can also be entered using the 0 through 9 number keys. Position the cursor on the appropriate digit and press the applicable number key. The cursor then moves right to the next least significant digit. For example, to display reading #0236, position the cursor all the way to the left (which is the most significant digit, or MSD) and press 0, 2, 3, 6. Note that if you enter a number that exceeds the buffer size, the reading at the highest memory location is displayed.

To exit from the data store recall mode, press the **EXIT** key.

Buffer statistics

With the data store in the recall mode, buffer statistics are displayed by using the DISPLAY TOGGLE key. Use the DISPLAY TOGGLE key to sequence through the statistics and return the Model 6482 to the normal data store recall state. Pressing the **EXIT** key at any time returns the instrument to the normal display state.

NOTE   *Statistics calculations will be performed on the presently selected function (MSR1, MSR2, RATIO, or DELTA). You can display these different functions while the statistics mode is active by pressing the appropriate function key. For example, if V/I readings are displayed on MSR1, all buffer statistics displayed are based on those readings.*

Minimum and maximum

This mode displays the minimum and maximum readings stored in the buffer. The buffer location number and timestamp are also provided for these readings. Go to the specific buffer location to obtain more data about the readings.

Peak-to-peak

This mode displays the peak-to-peak reading (peak-to-peak = maximum - minimum).

Average

The average mode displays the mean (average) of all measured readings stored in the buffer. The following equations are used to calculate mean:

$$z = \sum_{i=1}^{n} X_i$$

$$y = \frac{z}{n}$$

Where: y = The mean (average).
       z = The sum of the stored readings.
       X_i = A stored reading.
       n = The number of stored readings.

Standard deviation

This mode displays the standard deviation of buffered readings. The following equation is used to calculate standard deviation:

$$y = \sqrt{\dfrac{\displaystyle\sum_{i=n-1}^{n} X_i^2 - \left(\dfrac{1}{n} \left(\sum_{i=1}^{n} X_i \right) \right)^2}{n-1}}$$

Where: y = The average.

X_i = A stored reading.

n = The number of stored readings.

Timestamp format

Buffer readings can be recalled using the absolute timestamp format or the delta format. For the absolute format, readings are referenced to zero (0) seconds. For the delta format, the timestamp indicates the time between the displayed reading and the previous reading.

Perform the following steps to set the timestamp format:

1. While in the normal display mode, press the **CONFIG** key and then the **STORE** key to display the timestamp choices.
2. Place the cursor on **ABSOLUTE** or **DELTA** and press the **ENTER** key.

Timestamp accuracy

Because of internal timing methods, the timestamp value is only approximate. The method in which the timestamp is implemented limits its use in time-critical applications. If accurate test timing is crucial, it is recommended that an external timer be used in conjunction with the Model 6482.

The timestamp is based on an oscillator with a frequency of approximately 8 kHz. This oscillator is used as the system clock and is divided by eight to generate system "ticks" every millisecond. Therefore, the timestamp should provide 1 ms resolution for test timing. Because the actual oscillator frequency is 8.192 kHz (instead of 8.000 kHz), a system tick occurs every 8.192 kHz/8, or 1,024 times a second, which results in a system tick every 0.9765625 ms. As a result, the reported timestamp value deviates by 24 ms every second. Thus, to obtain more accurate timestamp values, multiply the timestamp displayed on the front panel or returned through a remote interface by a factor of 0.9765625.

Buffer considerations

From the front panel, 3000 source and measure readings can be stored and accessed using the method described earlier in this section. Over the bus, however, there are two separate 3000-reading buffers, for a total of 6000 readings. The :TRACe buffer is a 3000-reading buffer used by front-panel data store, bus :TRACe commands, and to store sweep data. The READ? buffer is a separate 3000-reading buffer that can only be accessed over the bus using the :READ? command (or alternatively by using an :INITiate followed by :FETCh? command). As described below, you can store and access data from these two buffers separately.

Using :TRACe commands to store data

Use :TRAC:POIN <n> and :TRIG:COUN <n> followed by :TRAC:FEED:CONT NEXT to store data (n = Number of readings; 3000 maximum). Use FORM:ELEM:TRAC CURR1 | CURR2 to select the channel. Turn on the output with :OUTP1 ON or :OUTP2 ON and then send :INIT to take the instrument out of idle and store readings. After data is stored, send :TRAC:DATA? to access it.

See Table 6-1 in this section for a summary of these commands (see Section 16, "TRACe subsystem," for more details).

Using :READ? to store data

Use :TRIG:COUN <n> to set the number of readings to be stored (n = Number of readings; 3000 maximum). Use the :FORM:ELEM CURR1 | CURR2 to select the channel. Turn on the output with :OUTP1 ON or :OUTP2 ON and then send the :READ? command to trigger and access readings. Once you access these readings, you will still be able to access previously stored :TRACe buffer readings using :TRAC:DATA?. See Section 8 and Section 16, "Trigger subsystem," for triggering details, and Section 15 for information on the :READ? command.

Remote command data store

This topic describes basic data store (buffer) capabilities as accessed over a remote interface.

Data store commands

Table 6-1 summarizes commands associated with data store operation. See Section 16, "TRACe subsystem," "DISPlay subsystem," "FORMat subsystem," and "CALCulate8," for more detailed information on these commands.

Table 6-1
Data store commands

Command	Description
:TRACe:DATA?	Read contents of buffer.
:TRACe:CLEar	Clear buffer.
:TRACe:FREE?	Read buffer memory status.
:TRACe:POINts <n>	Specify buffer size (n = Buffer size).
:TRACe:POINts:ACTual?	Query number of stored readings.
:TRACe:FEED:CONTrol <name>	Start or stop buffer. Name = NEXT (fill buffer and stop) or NEVer (disable buffer).
:TRACe:TSTamp:FORMat <name>	Select timestamp format. Name = ABSolute (reference to first buffer reading) or DELTa (time between buffer readings).
:CALCulate8:FORMat <name>	Select buffer statistic (name = MEAN, SDEViation, MAXimum, MINimum, or PKPK).
:CALCulate8:DATA?	Read buffer statistic data.
:FORMat:ELEMents:TRACe <name>	Select buffer data elements. Name = CURRent[1], CURRent2, CALCulate1, or CALCulate2, CALCulate3, CALCulate4, CALCulate5, CALCulate6, CALCulate7, TIME, STATus, ALL, or DEFault (CURRent[1] or CURRent2).
:DISPlay:MODE <name>	Select CALC8:DATA? source. Name = CALC3, CALC4, CALC5, or CALC6.

Data store programming example

Table 6-2 summarizes the commands for basic data store operation. These commands set up the Model 6482 as follows:

- Number of points: 10
- Trigger count: 10
- Data store feed: Channel 1 readings only
- Acquired data: All buffer readings, mean (average), and standard deviation.

NOTE   *See Appendix F for a complete program listing, which also shows how to determine when the buffer is full by reading the appropriate status register bits. See Section 13 for details on the status structure.*

Table 6-2
Data store example

Command	Description
`*RST`	Restore GPIB defaults.
`:SOUR1:VOLT 10`	Source 10 V.
`:TRAC:POIN 10`	Store 10 readings in data store buffer.
`:FORM:ELEM:TRAC CURR1`	Select channel 1 readings.
`:TRAC:FEED:CONT NEXT`	Enable the buffer.
`:TRIG:COUN 10`	Trigger count = Buffer points (10).
`:OUTP1 ON`	Turn on output #1.
`:INIT`	Trigger readings.
`:OUTP1 OFF`	Turn off output.
`:TRACE:DATA?`	Request all buffer readings.
`:DISP:MODE CALC3`	Select channel 1 for statistics.
`:CALC8:FORM MEAN`	Select mean buffer statistic.
`:CALC8:DATA?`	Request buffer mean data.
`:CALC8:FORM SDEV`	Select standard deviation statistic.
`:CALC8:DATA?`	Request standard deviation data.

Sweep Operation

In this section:

Sweep types

The Model 6482 can perform three basic sweep types:

- Linear staircase sweep
- Logarithmic staircase sweep
- Custom sweep

Linear staircase sweep

As shown in Figure 7-1, this sweep steps from a start voltage source value to an ending (stop) voltage source value. Programmable parameters include the start, stop, and step source voltage levels.

When this sweep is triggered to start, the output will go from the bias voltage level to the start source voltage level. The output will then change in equal steps until the stop source level is reached. With trigger delay set to zero (0), the time duration at each step is determined by the source delay and the time it takes to perform the measurement (NPLC setting). Note that the delay cannot change once a sweep is configured and running, and is the same for all steps.

Figure 7-1
Linear staircase sweep
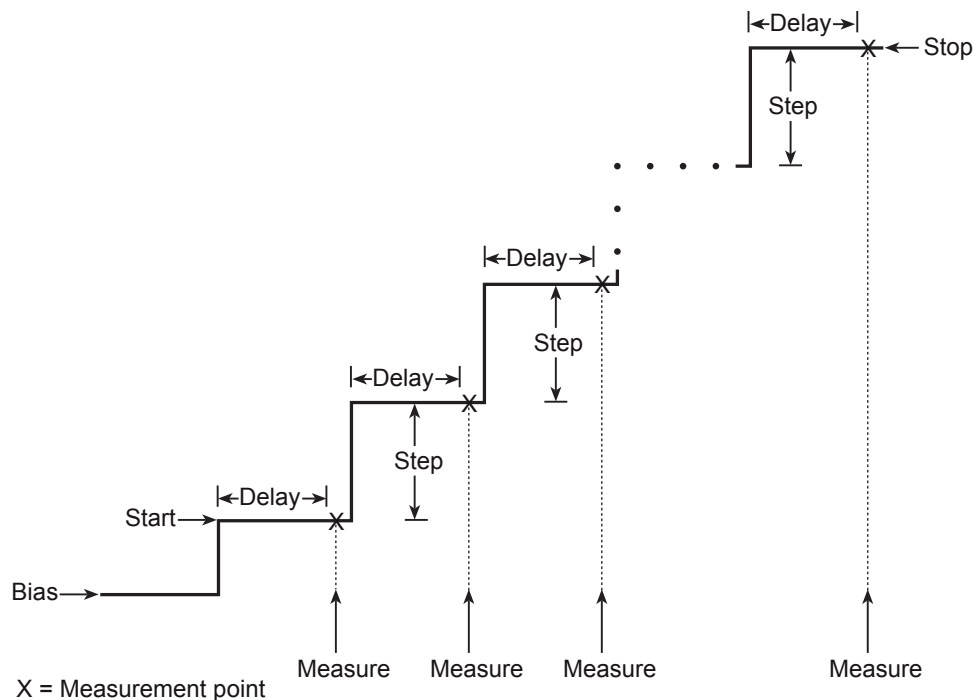
Logarithmic staircase sweep

This sweep is similar to the linear staircase sweep. The steps, however, are done on a logarithmic scale as shown in the example sweep in Figure 7-2. This is a 5-point logarithmic sweep from 1 to 10 V. As with the staircase sweep, the delay period is the same for all steps.

Figure 7-2
Logarithmic staircase sweep (example 5-point sweep from 1 to 10 volts)

X = Measurement point
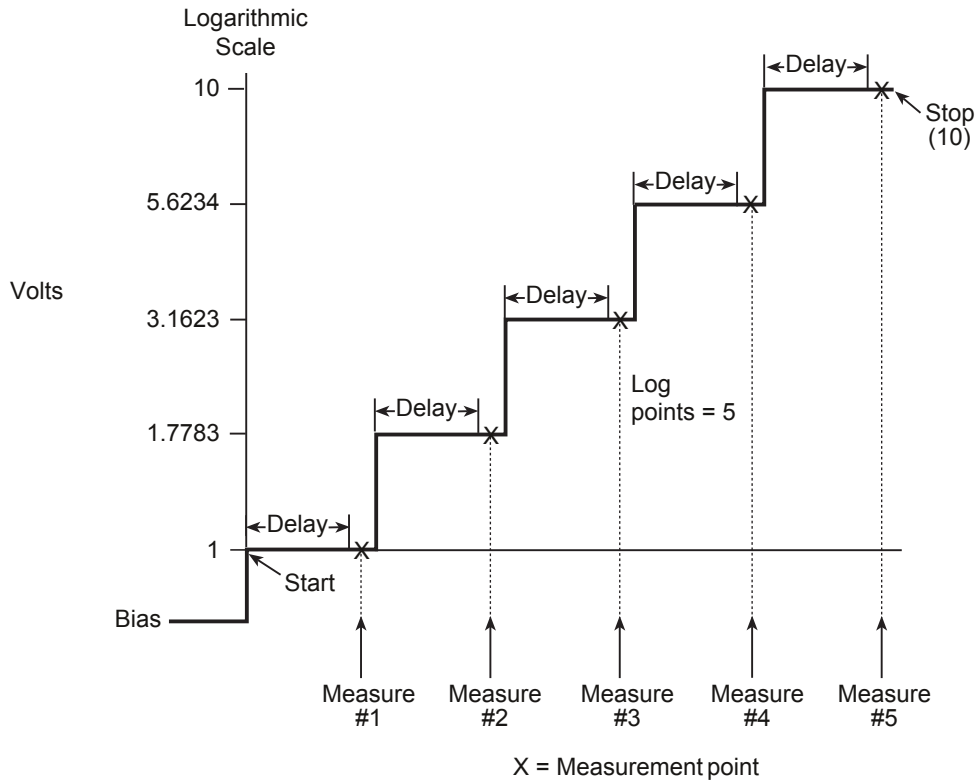
The programmable parameters for a logarithmic sweep include the start and stop levels and the number of measurement points for the sweep. The specified start, stop, and point parameters determine the logarithmic step size for the sweep. Step size for the sweep in Figure 7-2 is calculated as follows:

$$
\begin{aligned}
\text{logarithmic step size} &= \frac{\log_{10}(\text{stop}) - \log_{10}(\text{start})}{\text{points} - 1} \\
&= \frac{\log_{10}(10) - \log_{10}(1)}{5 - 1} \\
&= \frac{1 - 0}{4} \\
&= 0.25
\end{aligned}
$$

Thus, the five logarithmic steps for this sweep are 0, 0.25, 0.50, 0.75, and 1.00. The actual voltage bias levels at these points are listed in Table 7-1 (the voltage bias level is the antilogarithm of the logarithmic step).

Table 7-1
Logarithmic sweep points

Measure point	Logarithmic step	Bias level (volts)
Point 1	0	1
Point 2	0.25	1.7783
Point 3	0.50	3.1623
Point 4	0.75	5.6234
Point 5	1.0	10

When this sweep is triggered to start, the output will go from the bias level to the start source level (1 V) and sweep through the symmetrical logarithmic points. With trigger delay set to zero, the time duration at each step is determined by the source delay and the time it takes to perform the measurement (NPLC setting).

Custom sweep

This sweep type lets you configure a customized sweep. Programmable parameters include the number of measurement points in the sweep and the voltage source level at each point.

When this sweep is started, the output goes from the bias level to the first voltage bias and measure point in the sweep. The sweep will continue through the points in the order they were programmed and stop after the last voltage bias and measure point. With trigger delay set to zero (0), the time duration at each step is determined by the source delay and the time it takes to perform the measurement (NPLC setting). This delay is the same for all sweep points.

Custom sweep examples

The custom sweep can be configured to provide a 50 percent duty cycle pulse sweep. Figure 7-3 shows a pulse sweep that provides three 1 V pulses on a 0 V bias level. This pulse sweep is configured by specifying six points for the custom sweep. The specified voltage levels at points P0, P2, and P4 are 1 V, and the specified voltage levels at points P1, P3, and P5 are 0 V. Six measurements are performed for this sweep: Three at 1 V and three at 0 V.

Figure 7-3
Custom pulse sweep
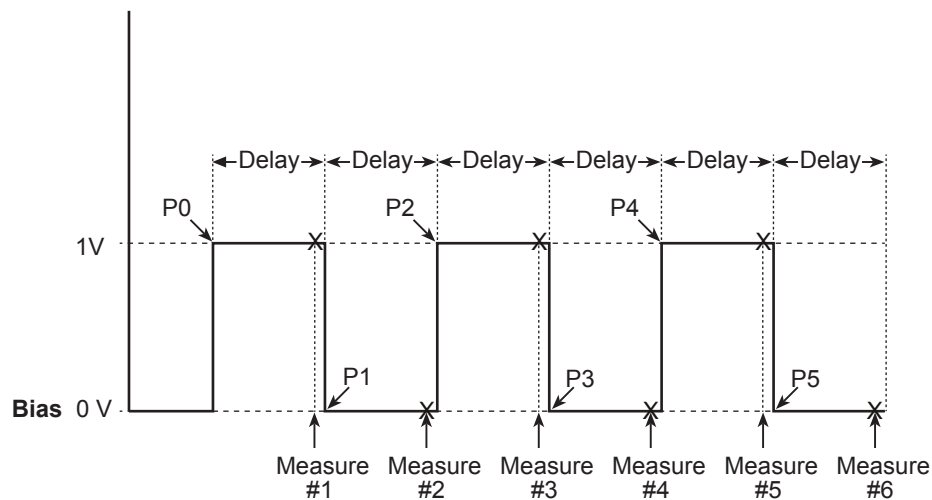
Figure 7-4 shows a custom sweep example with different pulse widths. In this example, the first two points are configured with the same source value so that the duration of the first pulse is effectively doubled.

Figure 7-4
Custom sweep with different pulse widths

All have same delay, but pulse widths differ.
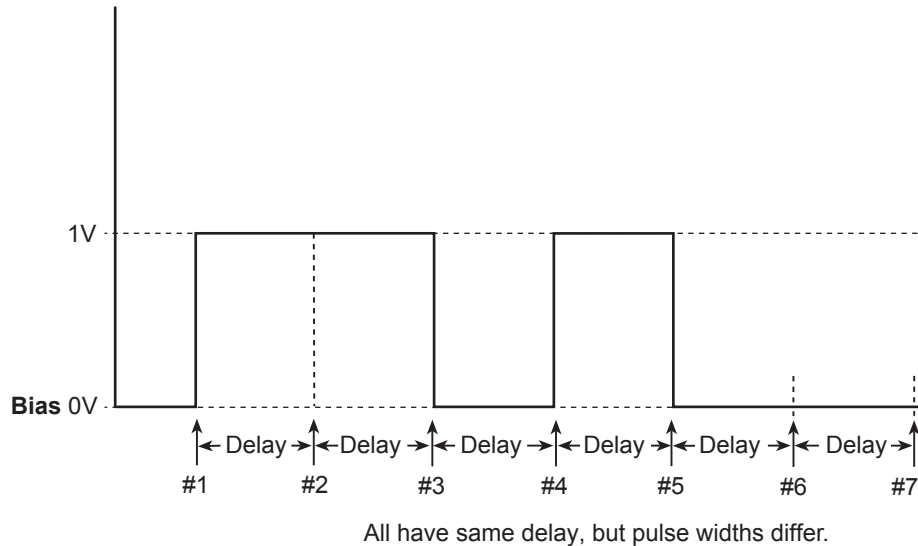
Configuring and running a sweep

This topic discusses the procedure for setting up and performing sweeps including selecting and configuring a sweep, setting the delay, and performing a sweep.

Front-panel sweep operation

Configuring a sweep

The sweep configuration menu is structured as described in the following paragraphs, and as shown in Figure 7-5. Note that bullets indicate the primary items of the sweep menu and dashes indicate the options of each menu item. Using Section 1, "Rules to navigate menus," go through the following menu to select and configure the sweep.

NOTE   *Before configuring a sweep, press SRC1 or SRC2 to select the channel to be used for the sweep. You can also switch between channel 1 and channel 2 by pressing SRC1 or SRC2 while in the CONFIGURE SWEEPS menu.*

CONFIGURE SWEEPS menu:

Press the **CONFIG** key and then the **SWEEP** key to display the sweep configuration menu.

- • **TYPE**: Use this menu item to select the type of sweep:
  - – NONE – Disables all sweeps.
  - – STAIR – When the linear staircase sweep is selected, you will be prompted to enter the START, STOP, and STEP levels.
  - – LOG – When the logarithmic staircase sweep is selected, you will be prompted to enter the START and STOP levels and to specify the number of measurement points.
  - – CUSTOM – With the custom sweep selected, you specify the number of measurement points (#-POINTS) in the sweep and the source level at each point (ADJUST- POINTS).

With the INIT option, you can set a consecutive range of measurement points in the sweep to a specific level. For example, assume that for a 20-point custom voltage sweep (# POINTS = 20), you want points 10 through 15 to be set for 1 V. After selecting the INIT option, set the VALUE to +1.000000 V, set the START PT to 10, and set the STOP PT to 15.

- **SWEEP-COUNT**: Use this menu item to specify how many sweeps to perform:
    – CUSTOM – With the custom sweep selected, you specify the number of measurement points (#-POINTS) in the sweep and the source level at each point (ADJUST- POINTS).
    – FINITE – Use this option to enter a discrete number of sweeps to perform with the results stored in the data store buffer. The maximum number of finite sweeps that can be performed is determined as follows:
    Maximum finite sweep count = 3000 / # points in sweep
    – INFINITE – Select this option to continuously repeat the configured sweep. Use the EXIT key to stop the sweep. Data is not stored in the buffer.
- **SOURCE-RANGING**: Use this menu item to control voltage bias source ranging:
    – BEST-FIXED – With this option, the Model 6482 will select a single fixed source range that will accommodate all of the source levels in the sweep. For example, if the minimum and maximum source levels in the sweep are 1 V and 30 V, the 30 V source range will be used.
    – AUTO-RANGE – With this option, the Model 6482 will select the most sensitive source range for each source level in the sweep. For example, for a 1 V source level, the 2 V source range will be used, and for a 3 V source level, the 30 V source range will be used. Note that the range changing process of AUTO-RANGE may cause transients in the sweep. If these transients cannot be tolerated, use the BEST- FIXED source range.
    – FIXED – With this option, the source remains on the range presently on when the sweep is started. For sweep points that exceed the source range capability, the source will out-put the maximum level for that range. For example, if the source is on the 2 V range when the sweep is started, it will remain on the 2 V range for the entire sweep. If the con-figured sweep points are 1 V, 2 V, 3 V, 4 V, and 5 V, the sweep will be 1 V, 2 V, 2.1 V, 2.1 V, and 2.1 V.

Figure 7-5
Sweep configuration menu tree
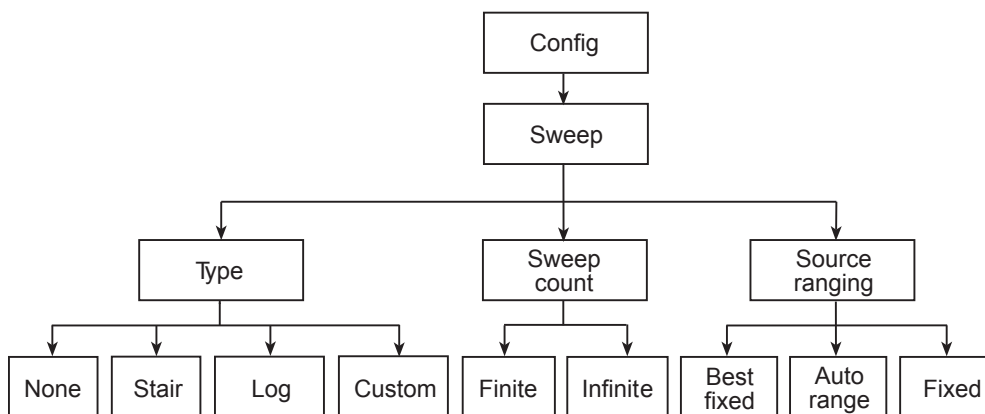
Setting delay

Generally, the time duration spent at each step (or point) of a sweep consists of the source delay and the time it takes to perform the measurement (NPLC setting).

The source delay is part of the source-delay-measurement (SDM) cycle and is used to allow the source to settle before the measurement is made. See Section 9, "Source-delay-measure cycle," for details.

The total time period of the source delay could include an autodelay and a user programmed delay. With autodelay enabled, 1 ms of delay is used. The user-programmable source delay adds 0000.0000 to 9999.9980 seconds of delay. See Section 3, "Source delay," to set these delays.

Additional delay for a sweep is available by using the trigger delay. This user-specified delay (0000.0000 to 9999.9990 seconds) occurs before each SDM cycle (device action) of the sweep. The trigger delay is executed before each new source-point in the sweep. See Section 8, "Front-panel operation of the trigger model" and "Configuring triggering," to set trigger delay.

Trigger count and sweep points

The trigger count and number of sweep points should be either the same or multiples of one another. For example, with five sweep points and a trigger count of 10, the sweep will run twice. See Section 8 for details on trigger count.

Performing sweeps

Procedures for the various sweep types are covered below.

NOTE   *The following procedure assumes that the Model 6482 is already connected to the device under test (DUT) as explained in Section 2.*

Performing a linear staircase sweep

Step 1. Configure the channel.

Configure the Model 6482 for a specific channel as follows:

1. Select the source function by pressing the **SRC1** or **SRC2** key.
2. Set the appropriate source level value.
3. Press the **MSR1** or **MSR2** key to select the measurement channel, and then select the measurement range.

See Section 3, "Basic Operation," for more information.

The source level you set becomes the bias level for the sweep. When turned on, the output will maintain this bias level until the sweep is started. Typically, 0 V is used as the bias level.

If using a fixed measurement range, either make sure it can accommodate every measurement point in the sweep or use autoranging.

Step 2. Configure the sweep.

Configure the sweep as follows:

1. Press the **SRC1** or **SRC2** key to select the channel, and then press the **EXIT** key to leave edit mode.
2. Press the **CONFIG** key and then the **SWEEP** key.
3. Select **TYPE**, and then press the **ENTER** key.
4. Select **STAIR**, and then press the **ENTER** key.
5. At the prompts, enter the START, STOP, and STEP values.
6. From the CONFIGURE SWEEPS menu, select **SWEEP-COUNT**, and then press the **ENTER** key.

NOTE   *The Model 6482 limits the number of sweeps (sweep-count) according to the number of steps required for each sweep. The maximum number of finite sweeps that can be performed is determined as follows: Maximum finite sweep count = 3000 / # points in sweep.*

7. Choose **FINITE** or **INFINITE**.
8. From the CONFIGURE SWEEPS menu, select **SOURCE-RANGING**, press the **ENTER** key, and then select **BEST-FIXED**, **AUTO-RANGE**, or **FIXED**.
9. Press the **EXIT** key to return to the normal display.

Step 3. Set the delay.

Set the source delay as follows:

1. Press the **CONFIG** key, and then the **SRC1** or **SRC2** key depending on the selected source channel.
2. Select **DELAY**, and then press the **ENTER** key.
3. Set the delay value, and then press the **ENTER** key.

Step 4. Turn the output on.

Press the **ON/OFF OUTPUT** key to turn the output on (blue OUTPUT indicator turns on). The Model 6482 will output the programmed bias level.

Step 5. Run the sweep.

To run the sweep, press the **SWEEP** key. After the sweep is completed, turn the output off by pressing the **ON/OFF OUTPUT** key.

Step 6. Read the buffer.

Press the **RECALL** key to access the readings stored in the buffer. Press the **DISPLAY TOGGLE** key to display statistical information. For additional information on statistical information, see Section 6, "Data Store."

Performing a logarithmic staircase sweep

Step 1. Configure the channel.

Configure the Model 6482 for the needed operations as follows:

1. Select the source channel by pressing the **SRC1** or **SCR2** key.
2. Set the source level.
3. Press **MSR1** or **MSR2** key to select the measurement channel, and then choose the measurement range.

Step 2. Configure the sweep.

Configure the sweep as follows:

1. Press the **SRC1** or **SRC2** key to select the channel, and then press the **EXIT** key to leave edit mode.
2. Press the **CONFIG** key and then the **SWEEP** key.
3. Select **TYPE**, and then press the **ENTER** key.
4. Select **LOG**, and then press the **ENTER** key.
5. At the prompts, enter values for **START**, **STOP**, and **NO-OF-POINTS**.

> *NOTE*   *The Model 6482 limits the number of sweeps (sweep-count) according to the number of steps required for each sweep. The maximum number of finite sweeps that can be performed is determined as follows: Maximum finite sweep count = 3000 / # points in sweep.*

6. From the CONFIGURE SWEEPS menu, select **SWEEP-COUNT**, press the **ENTER** key, and then select **FINITE** or **INFINITE**.
7. From the CONFIGURE SWEEPS menu, select **SOURCE-RANGING**, press the **ENTER** key, and then select **BEST-FIXED**, **AUTO-RANGE**, or **FIXED**.
8. Press the **EXIT** key to return to the normal display.

Step 3. Set the delay.

Set the source delay as follows:

1. Press the **CONFIG** key and then the **SRC1** or **SRC2** key, depending on the selected source channel.
2. Select **DELAY**, and then press the **ENTER** key.
3. Set the delay value, and then press the **ENTER** key.
4. Press **EXIT** key to return to the normal display.

Step 4. Turn the output on.

Press the **ON/OFF OUTPUT** key to turn the output on (blue OUTPUT indicator turns on). The Model 6482 will output the programmed bias level.

Step 5. Run the sweep.

To run the sweep, press the **SWEEP** key. After the sweep is completed, turn the output off by pressing the **ON/OFF OUTPUT** key.

Step 6. Read the buffer.

Press the **RECALL** key to access the readings stored in the buffer. Press the **DISPLAY TOGGLE** key to display statistical information.

Performing a custom sweep

Step 1. Configure the channel.

Configure the Model 6482 for the needed source-measure operations as follows:

1. Select the source channel by pressing the **SRC1** or **SRC2** key.
2. Set the source level.
3. Press the **MSR1** or **MSR2** key to select the measurement channel, and then select the measurement range.

Step 2. Configure the sweep.

Configure the sweep as follows:

1. Press the **SRC1** or **SRC2** key to select the channel, and then press the **EXIT** key to leave edit mode.
2. Press the **CONFIG** key and then the **SWEEP** key.
3. Select **TYPE**, and then press the **ENTER** key.
4. Select **CUSTOM**, and then press the **ENTER** key.
5. Use the displayed menu selections to enter the #-POINTS, individual point values (ADJUST-POINTS), and initial value (INIT).

NOTE   *The Model 6482 limits the number of sweeps (sweep-count) according to the number of steps required for each sweep. The maximum number of finite sweeps that can be performed is determined as follows: Maximum finite sweep count = 3000 / # points in sweep.*

6. From the CONFIGURE SWEEPS menu, select **SWEEP-COUNT**, press the **ENTER** key, and then select **FINITE** or **INFINITE**.

7. From the CONFIGURE SWEEPS menu, select **SOURCE-RANGING**, press the **ENTER** key, and then select **BEST-FIXED**, **AUTO-RANGE**, or **FIXED**.

8. Press the **EXIT** key to return to the normal display.

Step 3. Set the delay.

Set the source delay as follows:

1. Press the **CONFIG** key, and then depending on the selected source channel, press the **SRC1** or **SRC2** key.

2. Select **DELAY**, and then press the **ENTER** key.

3. Set the delay value, and then press the **ENTER** key.

4. Press the **EXIT** key to return to the normal display.

Step 4. Turn the output on.

Press the **ON/OFF OUTPUT** key to turn the output on (blue OUTPUT indicator turns on). The Model 6482 will output the programmed bias level.

Step 5. Run the sweep.

To run the sweep, press the **SWEEP** key. After the sweep is completed, turn the output off by pressing the **ON/OFF OUTPUT** key.

Remote sweep operation

Staircase sweep commands

Table 7-2 summarizes remote commands used for linear and logarithmic staircase sweep operation. See Section 16, "Configure sweeps," for more details on these commands.

Table 7-2
Linear and logarithmic staircase sweep commands

Command	Description
:SOURce[1]:VOLTage:MODE SWEep	Select channel 1 sweep mode.
:SOURce[1]:VOLTage:STARt <n>	Specify Channel 1 sweep start voltage (n = Voltage).
:SOURce[1]:VOLTage:STOP <n>	Specify Channel 1 sweep stop voltage (n = Voltage).
:SOURce[1]:VOLTage:STEP <n>	Specify Channel 1 sweep step voltage (n = Voltage).
:SOURce[1]:VOLTage:CENTer <n>	Specify Channel 1 sweep center voltage (n = Voltage).
:SOURce[1]:VOLTage:SPAN <n>	Specify Channel 1 sweep span voltage (n = Voltage).
:SOURce[1]:SWEep:RANGing <name>	Select Channel 1 source ranging (name = BEST, AUTO, or FIXed).
:SOURce[1]:SWEep:SPACing <name>	Select Channel 1 staircase sweep type (name = LINear or LOGarithmic).
:SOURce[1]:SWEep:POINts <n>	Set Channel 1 number of sweep points (n = points).
:SOURce[1]:SWEep:DIRection <name>	Set Channel 1 sweep direction. Name = UP (sweep start to stop) or DOWN (sweep stop to start).
:SOURce2:VOLTage:MODE SWEep	Select channel 2 sweep mode.
:SOURce2:VOLTage:STARt <n>	Specify Channel 2 sweep start voltage (n = Voltage).
:SOURce2:VOLTage:STOP <n>	Specify Channel 2 sweep stop voltage (n = Voltage).
:SOURce2:VOLTage:STEP <n>	Specify Channel 2 sweep step voltage (n = Voltage).
:SOURce2:VOLTage:CENTer <n>	Specify Channel 2 sweep center voltage (n = Voltage).
:SOURce2:VOLTage:SPAN <n>	Specify Channel 2 sweep span voltage (n = Voltage).
:SOURce2:SWEep:RANGing <name>	Select Channel 2 source ranging (name = BEST, AUTO, or FIXed).
:SOURce2:SWEep:SPACing <name>	Select Channel 2 staircase sweep type (name = LINear or LOGarithmic).
:SOURce2:SWEep:POINts <n>	Set Channel 2 number of sweep points (n = points).
:SOURce2:SWEep:DIRection <name>	Set Channel 2 sweep direction. Name = UP (sweep start to stop) or DOWN (sweep stop to start).

Staircase sweep programming example

As an example of linear staircase sweep operation, assume the Model 6482 will be used to generate the reverse-biased V-I characteristics of a device under test (DUT). For the purposes of this test, assume the following basic sweep parameters:

Source and measure channel: Channel 2
Source mode: Sweep
Start voltage: 1 V
Stop voltage: 10 V
Step voltage: 1 V
Source delay: 100 ms

Table 7-3 lists the command sequence for this programming example. See Section 2, "Connections," for details on how to connect the DUT to the channel 2 INPUT and OUTPUT connectors.

NOTE   *See Appendix F for a complete program listing.*

Table 7-3
Staircase sweep programming example

Command	Description
`*RST`	Restore GPIB default conditions.
`:FORM:ELEM CURR2`	Select channel 2 measurement data.
`:SENS2:CURR:RANG:AUTO ON`	Enable channel 2 measure autorange.
`:SOUR2:VOLT:STAR 1`	1 V channel 2 start voltage.
`:SOUR2:VOLT:STOP 10`	10 V channel 2 stop voltage.
`:SOUR2:VOLT:STEP 1`	1 V channel 2 step voltage.
`:SOUR2:VOLT:MODE SWE`	Select channel 2 sweep mode.[1]
`:SOUR2:SWE:RANG AUTO`	Channel 2 automatic source ranging.
`:SOUR2:SWE:SPAC LIN`	Select channel 2 linear staircase sweep.
`:TRIG:COUN 10`	Trigger count = Number of sweep points.[2]
`:SOUR2:DEL 0.1`	100 ms source delay.
`:OUTP2 ON`	Turn on channel 2 source output.
`:READ?`	Trigger sweep, request data.
`:OUTP2 OFF`	Turn off channel 2 source output.

[1] Send this command after START, STOP, and STEP to avoid delays caused by the rebuilding of the sweep when each command is sent.
[2] For a single sweep, the trigger count should equal number of points in sweep: Points = (Stop-Start)/Step + 1. You can use the SOUR:SWE:POIN? query to read the number of points.

Custom sweep commands

Table 7-4 summarizes remote commands used for custom sweep operation. See Section 16, "Configure list," for more details on these commands.

Table 7-4
Custom sweep commands

Command	Description
`:SOUR[1]:VOLT:MODE LIST`	Select channel 1 voltage list (custom) sweep mode.
`:SOUR[1]:LIST:VOLT < list>`	Define Channel 1 V-source list (list = V1, V2, … Vn).
`:SOUR[1]:LIST:VOLT:APP <list>`	Add Channel 1 V-source list values (list = V1, V2, …Vn).
`:SOUR[1]:LIST:VOLT:POIN?`	Query length of Channel 1 V-source list.
`:SOUR[1]:SWE:RANG <name>`	Select Channel 1 source ranging (name = BEST, AUTO, or FIXed).
`:SOUR2:VOLT:MODE LIST`	Select channel 2 voltage list (custom) sweep mode.
`:SOUR2:LIST:VOLT < list>`	Define Channel 2 V-source list (list = V1, V2, … Vn).
`:SOUR2:LIST:VOLT:APPend <list>`	Add Channel 2 V-source list values (list =V1, V2, …Vn).
`:SOUR2:LIST:VOLT:POINts?`	Query length of Channel 2 V-source list.
`:SOUR2:SWE:RANG <name>`	Select Channel 2 source ranging (name = BEST, AUTO, or FIXed).

Custom sweep programming example

As an example of custom sweep operation, assume a five-point sweep with the following parameters:

Source and measure channel: Channel 1
Voltage sweep mode: List (custom sweep)
Sweep voltage points: 7 V, 1 V, 3 V, 8 V, 2 V
Source delay: 100 ms

Table 7-5 summarizes the basic remote command sequence for performing the custom sweep described above.

Table 7-5
Custom sweep programming example

Command	Description
`*RST`	Restore GPIB default conditions.
`:FORM:ELEM CURR1`	Select channel 1 measurement data.
`:SOUR1:VOLT:MODE LIST`	Channel 1 list volts sweep mode.
`:SOUR1:LIST:VOLT 7,1,3,8,2`	7 V, 1 V, 3 V, 8 V, and 2 V Channel 1 sweep points.
`:TRIG:COUN 5`	Trigger count = Number of sweep points.
`:SOUR1:DEL 0.1`	100 ms Channel 1 source delay.
`:OUTP1 ON`	Turn on Channel 1 source output.
`:READ?`	Trigger sweep, request data.
`:OUTP1 OFF`	Output 1 off.

This page left blank intentionally.

In this section:

Front-panel operation of the trigger model

The flow chart in Figure 8-1 summarizes triggering for front-panel operation. The trigger model is modeled after the remote commands used to control triggering (refer to "Trigger model"). Key trigger model settings are included in The flow chart graphically illustrates front-panel operation of the trigger model. Note that the BENCH defaults are denoted by the "+" symbol.

The primary actions of the trigger model are source, delay, and measure. The source action outputs the programmed voltage bias value, and the programmed delay provides a settling period for the voltage bias source before the measurement is performed.

The trigger model consists of two layers (arm layer and trigger layer) to provide versatility. Programmable counters allow operations to be repeated, and various input and output trigger options are available to provide source-measure synchronization between the Model 6482 and other instruments using the trigger link.

Unless noted, the programmable aspects of the trigger model are performed from the CONFIGURE TRIGGER menu. See "Configuring triggering," page Section 8-12.

NOTE   *The trigger model and associated settings and commands affect both channels simultaneously.*

Idle

The Model 6482 is in idle mode when it is not operating in the arm layer or trigger layer of the trigger model. When in idle mode, the ARM indicator is off. To take the Model 6482 out of idle mode, turn the output ON.

The Model 6482 can be returned to an idle mode state at any time by selecting the HALT menu item of the CONFIGURE TRIGGER menu. See "Configuring triggering."

Figure 8-1
Front-panel operation of the trigger model
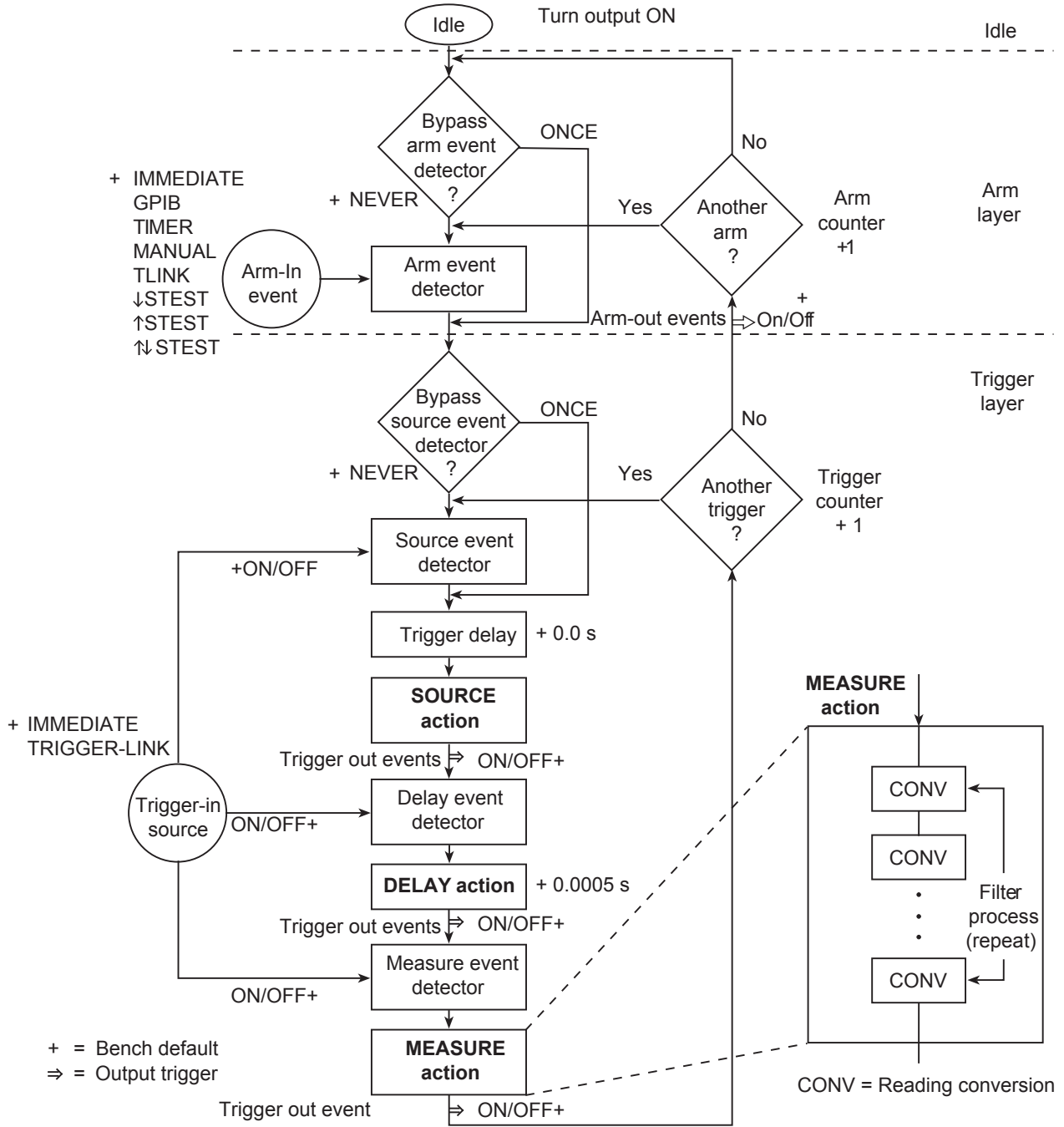
Event detection

Operation through the trigger model is delayed at the event detector until the programmed trigger event occurs. If an event detector has a bypass, operation can be programmed to loop around the event detector.

Arm layer

Bypass arm event detector: As shown in Figure 8-1, there is a bypass for the arm event detector. This bypass can only be used if TLINK or STEST is the selected arm-in event. The bypass serves to allow operation to bypass waiting for an event. With the event detector bypass set to ONCE, operation will loop around the arm event detector when the output is turned ON.

The programmable arm-in events for the arm layer are described as follows:

IMMEDIATE: Event detection occurs immediately, allowing operation to continue.

GPIB: Event detection occurs when a bus trigger (GET or *TRG) is received.

TIMER: With the timer selected, event detection occurs immediately when the output is turned ON. On repeated passes through the "Another Arm ? Yes" decision block and event detection occurs when the programmed timer interval expires. If operation takes the "Another Arm ? No" route, the timer is reset and event detection is allowed to occur immediately.

MANUAL: Event detection occurs when the TRIG key is pressed.

TLINK: Event detection occurs when an input trigger using the trigger link input line is received (see "Trigger link" for more information). With TLINK selected, you can loop around the arm event detector by setting the event detector bypass to ONCE.

↓**STEST:** Event detection occurs when the start of test (SOT) line of the OUTPUT ENABLE connector is pulsed low.

↑**STEST:** Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed high.

↑↓**STEST**: Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed either high or low.

NOTE   *STEST can be used only at the beginning of a sweep; STEST should not be used to trigger each point in a sweep. See "Input (SOT) line" on page 11-2 for more information.*

Trigger layer

The trigger layer uses three event detectors; one for each action (source, delay, and measure).

Bypass source event detector: As shown in Figure 8-1, there is a bypass for the source event detector. This bypass is in effect only if TRIGGER-LINK is the selected trigger-in source. With this event detector bypass set to ONCE, operation will proceed around the Source event detector.

The programmable trigger-in sources for the Trigger layer are described as follows:

IMMEDIATE: With IMMEDIATE selected, event detection for the three detectors is satisfied immediately. Operation proceeds through the Trigger layer to perform the source, delay, and measure actions.

TRIGGER-LINK: With TRIGGER-LINK selected, event detection at each enabled detector occurs when an input trigger using the trigger link input line is received. For example, if the TRIGGER-IN event for the Source event detector is ON, operation will pause at that detector until an input trigger is received. If, however, the Source event detector is disabled (OFF), operation will not pause. Operation will continue and perform the source action.

With the TRIGGER LINK TRIGGER-IN source selected, operation will go around the Source event detector (Figure 8-1) by setting the event detector bypass to ONCE.

Trigger delay

A programmable trigger delay is available before the source action. The trigger delay can be set manually from 0.00000 to 999.99990 seconds. Note that this delay is separate from the delay action of the source, delay, and measure (SDM) cycle. The delay action is discussed next.

Source, delay, and measure actions

The source, delay, and measure (SDM) cycle of the Model 6482 consists of three actions: Source, delay, and measure:

SOURCE action: Any programmed output voltage level changes are performed.

DELAY action: This programmable delay is used to allow the source to settle before a measurement is performed. It can be set manually from 0.00000 to 9999.99800 seconds, or autodelay can be enabled. With autodelay enabled, the Model 6482 automatically selects a nominal delay period based on the selected current measurement range.

NOTE   *The delay action is set from the CONFIGURE SRC1 or CONFIGURE SRC2 menu. See Section 3, "Source delay."*

MEASURE action: During this phase of the SDM cycle, the measurement process takes place. If the repeat filter is enabled (see the MEASURE action inset of Figure 8-1), the instrument samples the specified number of reading conversions to yield a single filtered reading (measurement). If using the moving filter or if the filter is disabled, only a single reading conversion will yield a reading.

Counters

Programmable counters are used to repeat operations within the trigger model layers. For example, if you are performing a 10-point sweep, set the trigger counter to 10. Operation will stay in the trigger layer until the 10 source-delay-measure points of the sweep are performed.

If you want to repeat the sweep three times, set the arm counter to three (3). Three 10-point sweeps can then be performed which equals 30 source-delay-measure actions.

The maximum buffer size for the Model 6482 is 3000 readings. The product of the two counter values cannot exceed 3000. For example, if you set an arm count of two (2), the maximum trigger count will be 1500 (3000 / 2 = 1500). However, you can set the arm count to INFINITE. With an infinite arm count, the maximum trigger count is 3000.

NOTE   *When a sweep is configured, the trigger model settings will not change until the sweep is started. After the sweep is finished, the trigger model will reset to the previous settings.*

Output triggers

The Model 6482 can be programmed to output a trigger (using the rear panel trigger link connector) after various trigger model operations. An output trigger is used to trigger another instrument to perform an operation. See "Trigger link," page Section 8-6, for more information.

Trigger Layer Output Triggers: After each action (source, delay, and measure), the Model 6482 can be programmed to send out an output trigger if TRIGGER-LINK is the selected trigger-in source. For example, if the trigger out event for measure is ON, an output trigger will be sent after the measure action. When used with a scanner, an output trigger after each measurement can signal the scanner to select the next channel in the scan.

Arm Layer Output Trigger: The Model 6482 can also be programmed to output a trigger when operation leaves the trigger layer and enters into the arm layer. This output trigger is typically sent to another instrument to signal the end of a scan or sweep.

Bench defaults

The bench defaults are listed as follows. They are also denoted in Figure 8-1 by the "+" symbol.

- Arm-In Event = IMMEDIATE
- Trigger-In Source = IMMEDIATE
- Arm Count = 1
- Trigger Count = 1
- Trigger Delay = 0.0 seconds
- Delay Action = 0.0005 seconds
- Source trigger in event = On
- Delay trigger in event = Off
- Measure Trigger In Event = Off
- Trigger out events = All trigger out events are disabled (off)
- Arm out events = Off
- Event detection bypasses = NEVER (both layers)

When the output is turned ON, the Model 6482 will run in a continuous loop around the trigger model. After each measure action, operation will continue at the top of the trigger model. The Model 6482 can be returned to an idle state by turning the output OFF.

Operation summary

The trigger model is designed to make source and measure applications more versatile. Typically, it allows you to perform a specified number of measurements at differing source levels.

For example, assume you want to perform three measurements each at two different channel 1 voltage bias source levels (1 V and 2 V). To do this, set the arm count to two, the trigger count to three (3), select SRC1, and use a 6-point custom sweep configured as follows:

P0000 = 1 V
P0001 = 1 V
P0002 = 1 V
P0003 = 2 V
P0004 = 2 V
P0005 = 2 V

When the sweep is started, operation falls into the Trigger Layer and performs three measurements at the 1 V source level. Operation then loops back into the Trigger Layer to perform three measurements at the 2 V source level. The six readings are stored in the buffer.

Note that after the sweep is finished, the Model 6482 does not return to an idle state. Operation continues at the top of the trigger model. Subsequent measurements are performed at the 2 V level and are not stored in the buffer.

For details on the custom sweep, see Section 7.

Trigger link

Input and output triggers are received and sent through the rear panel Trigger Link connector. The Trigger Link has four lines. At the factory, line #2 is selected for output triggers, and line #1 is selected for input triggers. These input/output line assignments can be changed from the CONFIGURE TRIGGER menu. See "Configuring triggering," page Section 8-12. The connector pinout is shown in Figure 8-2.

Figure 8-2
Rear panel trigger link pinout

TRIGGER LINK

Pin number	Description
1	Trigger Link 1
2	Trigger Link 2
3	Trigger Link 3
4	Trigger Link4
5	Trigger Link 5
6	Trigger Link 6
7	Ground
8	Ground
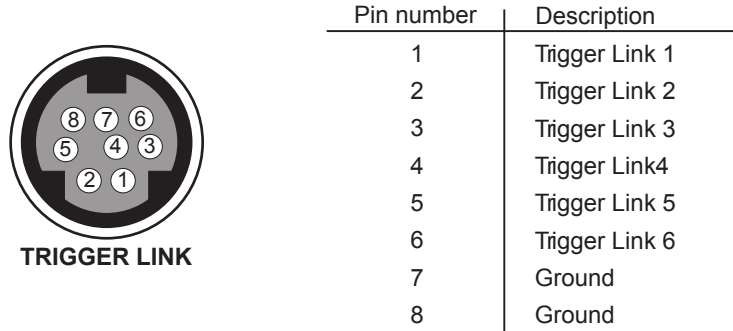
Input trigger requirements

An input trigger is used to satisfy event detection for a trigger model layer that is configured for the trigger link event. See "Front-panel operation of the trigger model." The input requires a falling-edge, TTL-compatible pulse with the specifications shown in Figure 8-3.

Figure 8-3
Trigger link input pulse specifications

Triggers on
leading edge

TTL high
(2 V-5 V)

TTL low
(-0.8 V)

10 μs
minumum
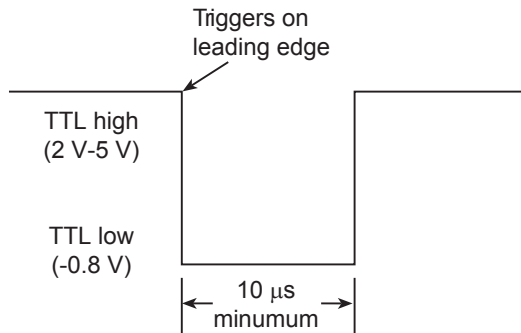
Output trigger specifications

The Model 6482 can be programmed to output a trigger after various trigger model actions. See "Front-panel operation of the trigger model." The output trigger provides a TTL-compatible output pulse that can be used to trigger other instruments. The specifications for this trigger pulse are shown in Figure 8-4. A trigger link line can source 1 mA and sink up to 50 mA.

Figure 8-4
Trigger link output pulse specifications

Meter complete

TTL High
(3.4 V Typical)

TTL Low
(0.25 V Typical)

10 μs
minumum
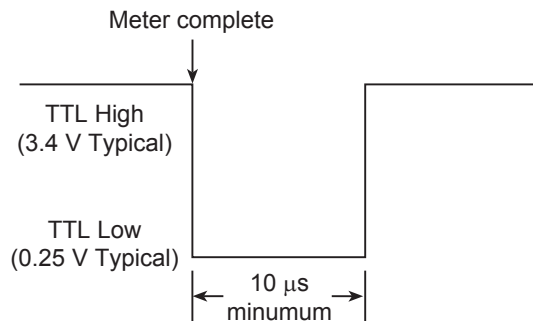
External triggering example

In a simple test system, you may want to close a switching channel and then measure the current of the device under test (DUT) connected to that channel. This test system is shown in Figure 8-5, which uses a Model 6482 to measure 10 DUTs switched by a Model 7011 multiplexer card in a Model 7001/7002 Switch System.

Figure 8-5
DUT test system
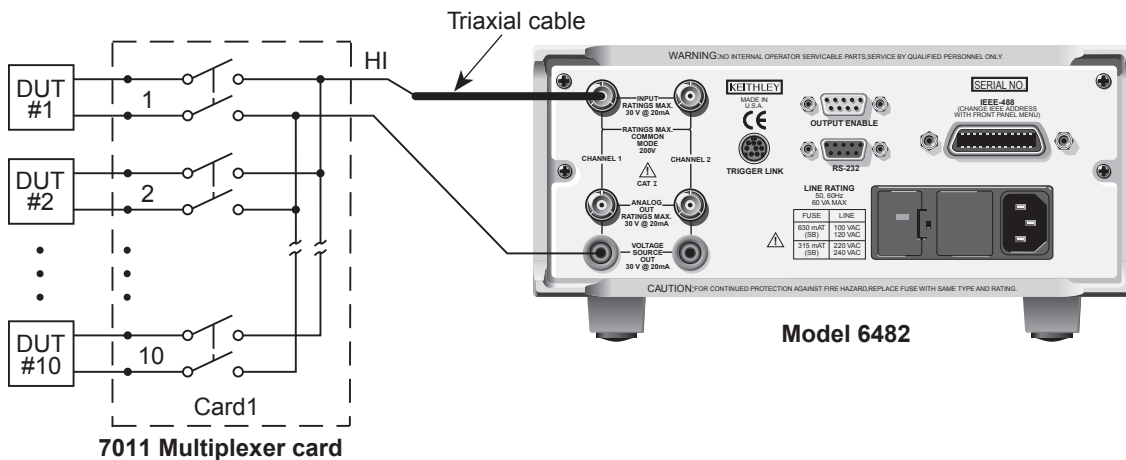
The trigger link connections for this test system are shown in Figure 8-6. The trigger link of the Model 6482 is connected to the trigger link (IN or OUT) of the switching mainframe. Note that with the default trigger settings of the switching mainframe, line 1 is an input, and line 2 is an output.

For this example, the Model 6482 and switching system (mainframe) are configured as follows.

Model 6482 setup

Step 1. Restore bench defaults.

1. Press the **MENU** key, select **SAVESETUP**, then press the **ENTER** key.
2. From the SAVESETUP menu, select **RESET**, then press the **ENTER** key.
3. Select **BENCH**, and then press the **ENTER** key.

Step 2. Configure channel.

1. Press the **MSR1** or **MSR2** key to select the channel to measure, and then select the range.
2. Press **SRC1** or **SRC2** to select the voltage source, and then set the source to the value using the EDIT or numeric keys.

Step 3. Set up trigger parameters.

1. Press the **CONFIG** key, and then the **TRIG** key to access the CONFIGURE TRIGGER menu.
2. Select **TRIG-LAYER** and then press the **ENTER** key to access the CONFIGURE TRIG-LAYER menu.

Step 4. Set trigger-in event to TRIGGER-LINK.

1. Select **TRIGGER-IN** and then press the **ENTER** key.
2. Select **TRIGGER-LINK** and then press the **ENTER** key.

Figure 8-6
Trigger link connections

Model 7001 or 7002 Switch System

Model 6482
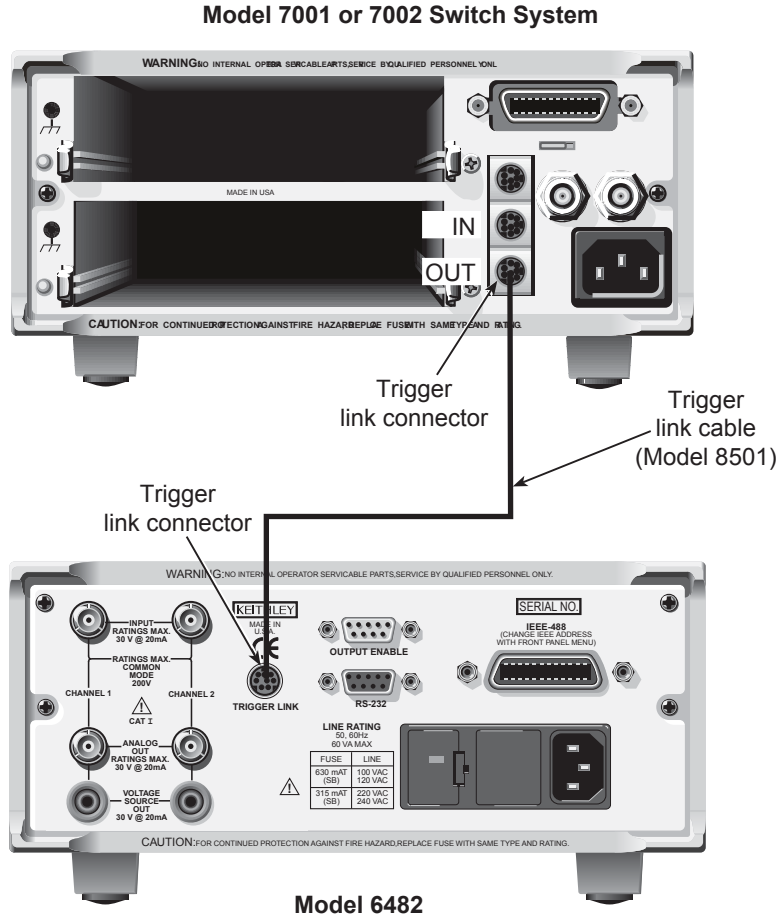
Step 5. Set trigger input line to #2.

Select **#2** and then press the **ENTER** key three times to return to the CONFIGURE TRIG-LAYER menu.

Step 6. Set trigger output line to #1.

1. Select **TRIGGER-OUT** and then press the **ENTER** key.
2. Select **LINE** and then press the **ENTER** key.
3. Select **#1** and then press the **ENTER** key.

Step 7. Set trigger out events to MEAS=ON (all others to OFF).

1. Select **EVENTS**, and then press the **ENTER** key.
2. Set MEAS=**ON** (toggle the value to **ON** using the EDIT ▲ key).
3. Make sure SOURCE=**OFF** and DELAY=**OFF** (set if necessary).
4. Press the **ENTER** key, and then press the **EXIT** key to return to the CONFIGURE TRIG-LAYER menu.

Step 8. Set trigger count to 10.

1. Press the **EDIT** ▶ key to scroll to the end of the menu selections and select **COUNT**, and then press the **ENTER** key.
2. Set the count to 10, and then press the **ENTER** key.

3. Press the **EXIT** key twice to leave the CONFIGURE TRIGGER menus.

Switching mainframe setup

Step 1. Restore bench defaults.

1. Press the **MENU** key, select **SAVESETUP**, and then press the **ENTER** key.
2. From the SAVESETUP menu, select **RESET**, and then press the **ENTER** key.
3. Press the **ENTER** key to confirm the action.
4. Press the **ENTER** key to return to the SETUP MENU.
5. Press the **EXIT** key to leave the SETUP MENU.
6. Press the **EXIT** key to leave the MAIN MENU.

Step 2. Set up scan list: 1!1 - 1!10.

1. Press the **SCAN LIST** key.
2. Press **1**, **1 - 1**, **1**, **0**, and then press the **ENTER** key.

Step 3. Set the number of scans to 1.

1. Press the **SCAN** key, select **SCAN-CONTROL**, and then press the **ENTER** key.
2. From the SCAN CONTROL menu, select **NUMBER-OF-SCANS**, and then press the **ENTER** key.
3. Select **ENTER-SCAN-COUNT**, and then press the **ENTER** key.
4. Set the count to **1**, and then press the **ENTER** key.
5. Press the **EXIT** key to return to the CONFIGURE SCAN menu.

Step 4. Set channel spacing to trigger-link.

1. Select **CHAN-CONTROL** from the CONFIGURE SCAN menu, and then press the **ENTER** key.
2. Select **CHANNEL-SPACING** from the CHANNEL CONTROL menu, and then press the **ENTER** key.
3. Select **TRIGLINK**, and then press the **ENTER** key.
4. Select **ASYNCHRONOUS**, and then press the **ENTER** key.
5. To leave the CONFIGURE SCAN menu, press the **ENTER** key twice, and then the **EXIT** key three times.

Operation

1. To store the readings in the Model 6482 buffer, press the **STORE** key and set the buffer size to 10. When the ENTER key is pressed, the asterisk (*) indicator will turn on to indicate the buffer is enabled. See Section 6 for details.
2. Turn the Model 6482 OUTPUT ON. The Model 6482 waits for an external trigger from the switching mainframe.
3. Press the **STEP** key on the Model 7001/7002 to take it out of the idle state and start the scan. The scanner's output pulse triggers the Model 6482 to take a reading and store it. The Model 6482 then sends a trigger pulse to the switching mainframe to close the next channel. This process continues until all 10 channels are scanned, measured, and stored.

Details of this testing process are explained in the following paragraphs and are referenced to the operation model shown in Figure 8-7.

Figure 8-7
Operation model for triggering example
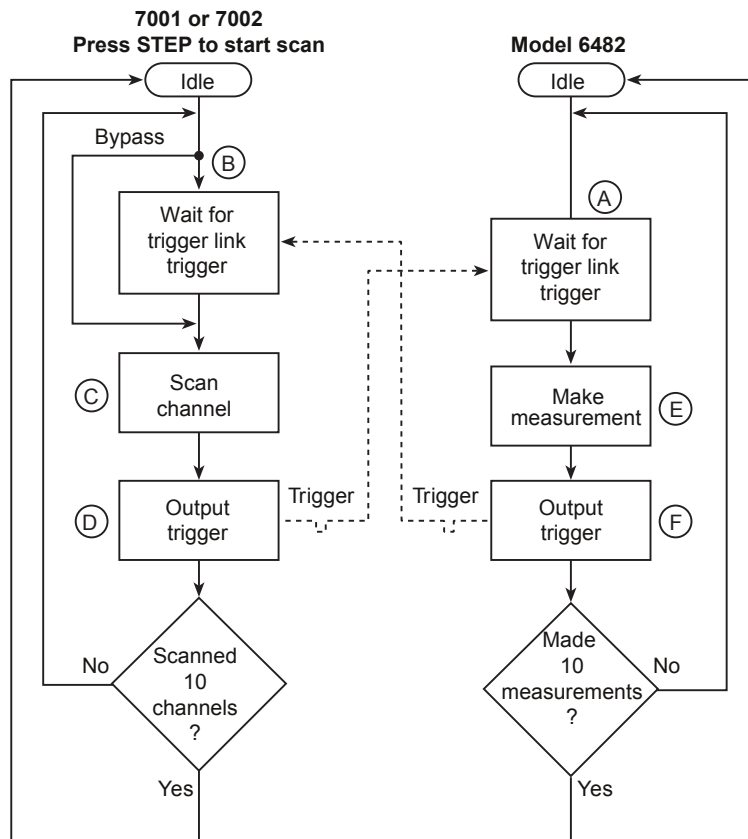
A) Turning the Model 6482 output on places it at point A in the flow chart, where it waits for an external trigger.

B) Pressing the **STEP** key takes the Model 7001/7002 out of the idle state and places operation at point B in the flow chart.

C) For the first pass through the model, the scanner does not wait at point B. Instead, it closes the first channel (point C).

D) After the relay settles, the Model 7001/7002 outputs a trigger pulse. Because the instrument is programmed to scan 10 channels, operation loops back to point B, where it waits for an input trigger.

E) and F) With the Model 6482 operation at point A, the output trigger pulse from the Model 7001/7002 triggers a measurement of device under test (DUT) #1 (point E). After the measurement is complete, the Model 6482 outputs a trigger pulse and then loops back to point A, where it waits for another input trigger.

The trigger applied to the Model 7001/7002 from the Model 6482 closes the next channel in the scan, which then triggers the Model 6482 to measure that device under test (DUT). This process continues until all 10 channels are scanned and measured.

Configuring triggering

Triggering is configured from the CONFIGURE TRIGGER menu. This menu is structured as follows.

NOTE   *See Front-panel operation of the trigger model, for details on the programmable aspects of triggering.*

CONFIGURE TRIGGER menu

Press the **CONFIG** key and then the **TRIG** key to display the menu shown below and in Figure 8-8. Note that bullets indicate the primary menu items; dashes and slashes indicate available options and attributes. See Section 1, "Rules to navigate menus," to check and change trigger options.

- **ARM-LAYER**: Use this menu item to configure the arm layer of the trigger model:
  - **ARM-IN**: Use to select the detection event for the arm layer:
    - **IMMEDIATE**: Event detection occurs immediately.
    - **MANUAL**: Event detection occurs when the TRG key is pressed.
    - **GPIB**: Event detection occurs when a bus trigger (GET or *TRG) is received.
    - **TIMER**: Initially, event detection is satisfied immediately. Subsequent event detection occurs after the timer interval elapses. After selecting this arm event, you will be prompted to specify the timer interval (in seconds).
    - **MANUAL**: Event detection occurs when the TRIG key is pressed.
    - **TLINK**: After selecting this arm event, you will be prompted to select the input line for the trigger link and the state of the event detection bypass. With ONCE selected, operation will loop around the arm event detector on each new pass through the trigger model. With NEVER selected, operation always waits for the input trigger.
    - ↓**STEST**: Event detection occurs when the start of test (SOT) line of the OUTPUT ENABLE connector is pulsed low. After selecting this arm event, you will be prompted to select the state of the event detection bypass. With ONCE selected, operation will loop around the arm event detector on each new pass through the trigger model. With NEVER selected, operation always waits for the input trigger.
    - ↑**STEST**: Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed high. After selecting this arm event, you will be prompted to select the state of the event detection bypass.
    - ↑↓**STEST**: Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed either high or low. After selecting this arm event, you will be prompted to select the state of the event detection bypass.
  - **ARM OUT**: Use to configure the arm layer output trigger:
    - **LINE**: Select the trigger link line for the output trigger: Line #1, #2, #3, #4, #5, or #6.
    - **EVENTS**: Enable (ON) or disable (OFF) the arm layer output triggers. TRIG-LAYER-DONE ON enables an output trigger when the trigger layer is exited.
  - **COUNT**: Specify the arm count, FINITE (programmable count) or INFINITE (non-stopping count).

- **TRIG-LAYER**: Use this menu item to configure the trigger layer of the trigger model:
  - **TRIGGER-IN**: Use to select the detection event for the trigger layer:
    - **IMMEDIATE**: Event detection occurs immediately.
    - **TRIGGER-LINK**: After selecting this trigger-in source, you will be prompted in sequence as follows:
      > TRIG-IN TLINK LINE: Select the input line (#1, #2, #3, #4, #5, or #6) for the trigger link.
      > EVENT-DETECT-BYPASS: Set the bypass for the Source Event Detector. With ONCE, operation will loop around the Source Event Detector. With NEVER selected, operation will wait for an input trigger.
      > TRIGGER-IN-EVENTS: Enable (ON) or disable (OFF) trigger-in events (SOURCE, DELAY, and MEAS). With a trigger-in event ON, operation will wait at that event for an input trigger. With the trigger-in event OFF, operation will not wait; it will continue and perform the appropriate action.
  - **TRIGGER OUT**: Use to configure the trigger layer output trigger:
    - **LINE**: Select the trigger link line for the output trigger; line #1, #2, #3, #4, #5, or #6.
    - **EVENTS**: Enable (ON) or disable (OFF) output triggers that occur after the SOURCE, DELAY, and MEAS actions.
  - **DELAY**: Specify the time delay (in seconds) for the trigger delay.
  - **COUNT**: Specify the trigger count.
- **HALT**: Use to return the Model 6482 to the idle state. HALT does not turn off the output. The programmed source level will still be available at the OUTPUT terminals. The following actions will take the Model 6482 out of an idle state:
  - Turn the output off and then on again.
  - Reselect the arm or trigger event.
  - Exit from the menu structure, and then reenter it by pressing CONFIG and then TRIG.

Figure 8-8
Configure trigger menu tree
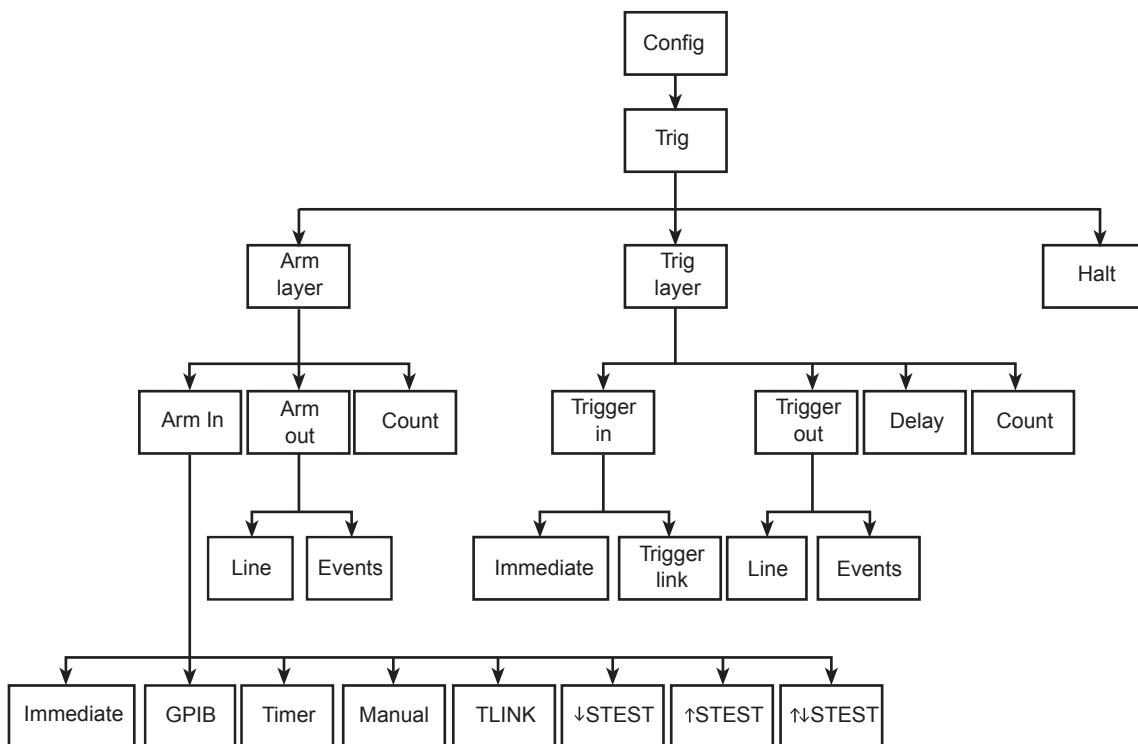
Remote operation of the trigger model

Trigger model

The trigger model flow chart in Figure 8-9 summarizes remote trigger operation. Operation is controlled by SCPI commands from the trigger subsystem. Key remote commands are included in the trigger model. Also note that the GPIB defaults are denoted by the "+" symbol.

The primary actions of the trigger model are source, delay, and measure. The source action outputs the programmed voltage or current value, and the programmed delay provides a settling period for the source before the measurement is performed.

The trigger model consists of two layers (Arm layer and Trigger layer) to provide versatility. Programmable counters allow operations to be repeated, and various input and output trigger options are available to provide source-measure synchronization between the Model 6482 and other instruments (using the trigger link).

Idle and initiate

The instrument is considered to be in the idle state (ARM indicator off) when it is not operating within the trigger model layers. While in the idle state, the instrument cannot perform any measurements. An initiate command is required to take the instrument out of an idle state. The following commands perform an initiate operation:

- :INITiate
- :READ?
- :MEASure?

Conversely, if the instrument is taking readings, most commands (except DCL, SDC, IFC, and ABORt) are queued and will not be executed until the instrument returns to the idle state.

The :MEASure? command will automatically turn the output on. Note that after the instrument returns to the idle state, the output will remain on.

When automatic output-off is enabled (:SOURce[1]1:CLEar:AUTO ON or :SOURce2:CLEar:AUTO ON), any of the three commands can be used to initiate operation (:INITiat, :READ?, or :MEASure?). The source output will automatically turn on at the beginning of each source-delay-measure (SDM) cycle and turn off after each measurement is completed.

While operating within the trigger model (ARM indicator on), most commands will not be executed until the Model 6482 completes all of its programmed source-measure operations and returns to the idle state. The IFC (interface clear), SDC (selected device clear) and DCL (device clear) commands can be executed under any circumstance while operating within the trigger model. The following commands will abort any other command or query.

- :ABORt
- :SYSTem:PRESet
- *TRG or GET
- *RST
- *RCL

NOTE   *SDC, DCL, or :ABORt place the Model 6482 in the idle state. For fastest response, use SDC or DCL to return to the idle state.*

Figure 8-9
Remote operation of the trigger model

* GPIB default parameter for TRIGGER:OUTPut is NONE
** SOURceX = SOURce[1] or SOURce2
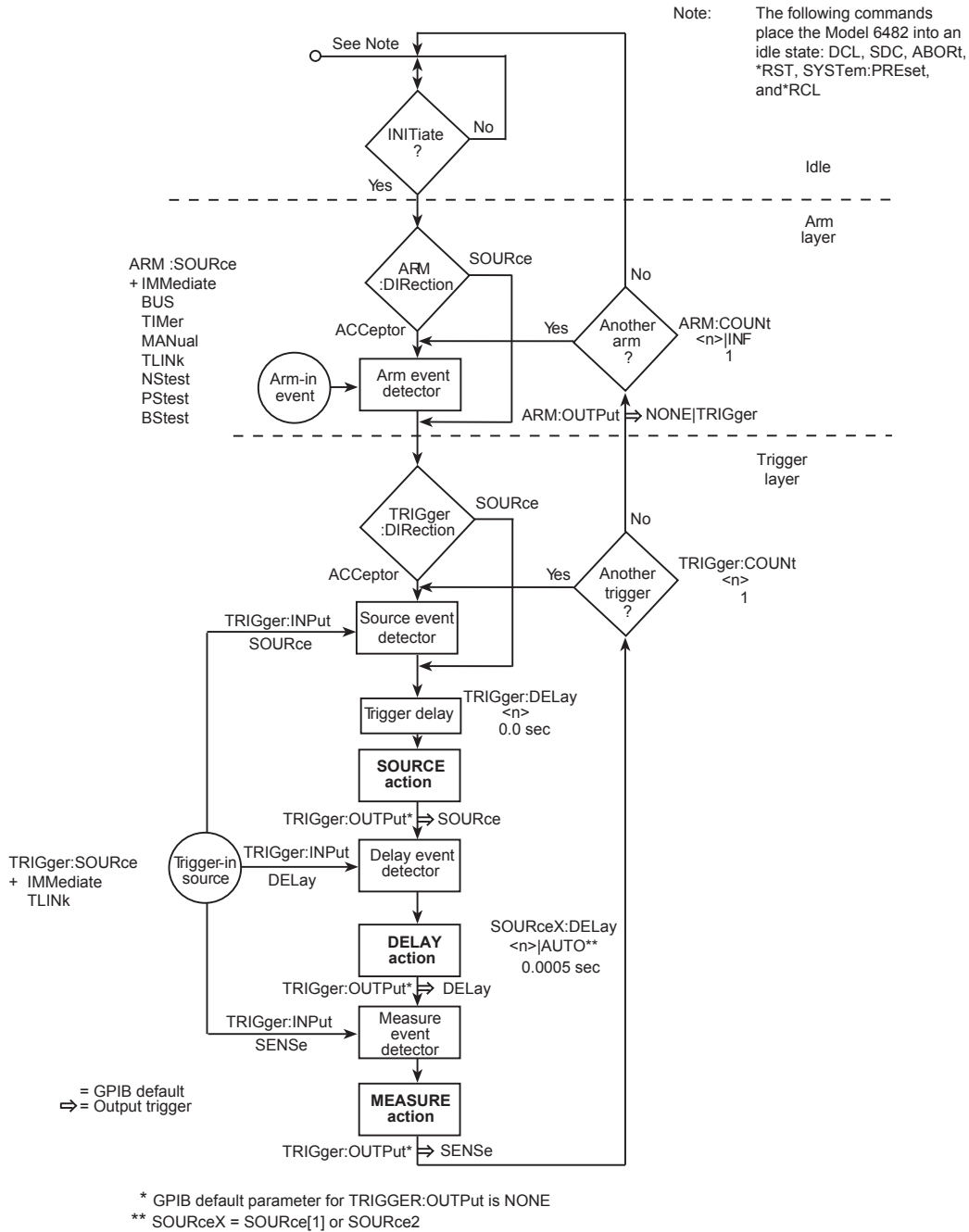
Event detection

Once the instrument is taken out of an idle state, operation proceeds through the trigger model to perform the source, delay, and measure actions.

Operation through the trigger model is delayed at the event detector until the programmed trigger event occurs. If an event detector has a bypass (:DIRection), operation can be programmed to loop around the event detector.

Arm layer

Arm detector event bypass: As shown in Figure 8-9, there is a bypass for the Arm Event Detector (ARM:DIRection). This bypass can only be used if TLINk, PSTest, NSTest, or BSTest is the selected arm-in event. The bypass serves to continue operation. With the bypass set to SOURce, operation will loop around the Arm Event Detector when an INITiate command is sent (assuming the output is turned ON).

The programmable arm-in events for the arm layer are described as follows:

IMMediate: Event detection occurs immediately, allowing operation to continue.

BUS: Event detection occurs when a bus trigger (GET or *TRG) is received.

TIMer: Event detection occurs immediately on the initial pass through the trigger model. Each subsequent detection is satisfied when the programmed timer interval elapses. The timer resets to its initial state when the instrument goes into an idle state.

MANual: Event detection occurs when the TRIG key is pressed. The Model 6482 must be in LOCAL mode for it to respond to the TRIG key. Press the LOCAL key or send GTL over the bus to take the Model 6482 out of remote operation.

TLINk: Event detection occurs when an input trigger using the trigger link input line is received (see "Trigger link" for more information). With TLINk selected, you can loop around the Arm Event Detector by setting the event detector bypass (ARM:DIRection) to SOURce.

NSTest: Event detection occurs when the start of test (SOT) line of the OUTPUT ENABLE connector is pulsed low.

PSTest: Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed high.

BSTest: Event detection occurs when the SOT line of the OUTPUT ENABLE connector is pulsed either high or low.

NOTE   *NSTest, PSTest, and BSTest can be used only at the beginning of a sweep and should not be used to trigger each point in a sweep. See "Input (SOT) line" on page 11-2 for more information.*

Trigger layer

The trigger layer uses three event detectors; one for each action (source, delay, and measure).

Source event detector bypass: As shown in Figure 8-9, there is a bypass (TRIGger:DIRection) for the source event detector. This bypass is in effect only if TLINk is the selected trigger-in source. With this event detector bypass set to SOURce, operation will proceed around the source event detector.

The programmable trigger-in sources for the trigger layer are described as follows:

IMMediate: With immediate selected, event detection for the three detectors is satisfied immediately. Operation proceeds through the trigger layer to perform the source, delay, and measure actions.

TLINk: With TLINk selected, event detection at each enabled detector occurs when an input trigger using the trigger link input line is received. A detector is enabled by including its parameter name with the TRIGger:INPut command.

For example, to enable the delay event detector and measure event detector, the following command must be sent:

```
TRIGger:INPut DELay, SENSe
```

The above command disables the source event detector because the parameter name (SOURce) is not included in the parameter list.

With the source event detector disabled, operation will not pause; it will continue and perform the source action. Operation will pause at the delay event detector until an input trigger is received, and then it will pause at the measure event detector until another input trigger is received.

Trigger delay

A programmable delay is available before the source action. The trigger delay can be set manually from 0.00000 to 999.99990 seconds. Note that this delay is separate from the delay action of the source, delay, and measure (SDM) cycle. The delay action is discussed in the following topic.

Source, delay, and measure actions

The SDM cycle of the Model 6482 consists of three actions: Source, delay, and measure:

SOURCE action: Any programmed output voltage level changes are performed.

DELAY action: This programmable delay is used to allow the source to settle before a measurement is performed. It can be set manually from 0.00000 to 9999.99800 seconds, or autodelay can be enabled. With autodelay enabled, the Model 6482 automatically selects a nominal delay period based on the selected current range.

MEASURE action: During this phase of the SDM cycle, the measurement process takes place. If the repeat filter is enabled, as shown in Figure 8-10, the instrument samples the specified number of reading conversions to yield a single filtered reading (measurement). If using the moving filter or if the filter is disabled, only a single reading conversion will yield a reading.

Figure 8-10
Measure action
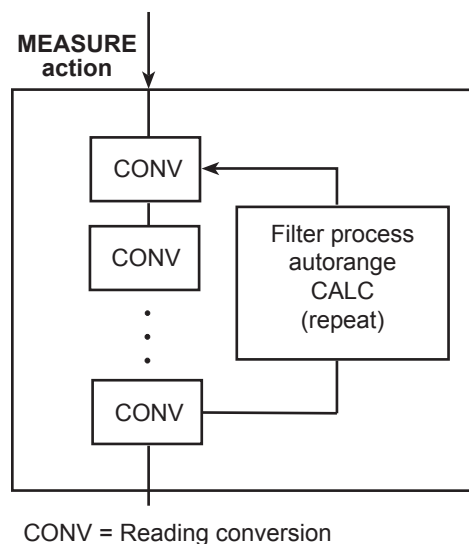
CONV = Reading conversion

Counters

Programmable counters are used to repeat operations within the trigger model layers. For example, if performing a 10-point sweep, the trigger counter would be set to 10 (TRIGger:COUNt 10). Operation will stay in the trigger layer until the 10 source-delay-measure points of the sweep are performed.

If you want to repeat the sweep three times, set the arm counter to three (ARM:COUNt 3). Three 10-point sweeps will then be performed (a total of 30 source-delay-measure actions).

The maximum buffer size for the Model 6482 is 3000 readings. The product of the finite values of the two counters cannot exceed 3000. For example, if you set an arm count of two, the maximum trigger count will be 3000 (3000 / 2 = 1500). However, you can set the arm count to infinite (INF). With an infinite arm count, the maximum trigger count is 3000.

NOTE   *With front-panel operation only, when a sweep is configured, the trigger model settings will not change until the sweep is started. After the sweep is finished, the trigger model will reset to the previous settings.*

Output triggers

The Model 6482 can be programmed to output a trigger (using the rear panel trigger link connector) after various trigger model operations. An output trigger is used to trigger another instrument to perform an operation. See "Trigger link," page Section 8-6, for more information.

Trigger layer output triggers: The Model 6482 can be programmed to output a trigger after each action of the SDM cycle (source, delay, and measure). Output triggers are controlled with the TRIGger:OUTPut command. For example, to output a trigger after the measure action, the following command must be sent:

```
TRIGger:OUTPut SENSe
```

The above command disables output triggers for the source and delay actions because their parameter names (SOURce and DELay) are not included in the parameter list. When used with a scanner, an output trigger after each measurement can signal the scanner to select the next channel in the scan.

Arm layer output trigger: As shown in Figure 8-9, the Model 6482 can be programmed to output a trigger when operation leaves the arm layer and enters the trigger layer. This output trigger is typically sent to another instrument to signal the end of a scan or sweep. The ARM:OUTPut command is used to control this output trigger. The TRIGger parameter enables the trigger when the trigger layer is exited, and the NONE parameter disables the output trigger.

GPIB defaults

The GPIB defaults are listed as follows. They are also denoted in Figure 8-9 by the "+" symbol.

- Arm-In Event = Immediate
- Trigger-In Source = Immediate
- Arm Count = 1
- Trigger Count = 1
- Trigger Delay = 0.0 seconds
- Delay Action = 0.001 seconds
- Enabled event detector = Source event detector (delay and measure detection disabled)
- Enabled output triggers = None
- Event detection bypasses = Acceptor (both layers)

With output turned ON (OUTPut1 ON or OUTPut2 ON), the Model 6482 will perform one SDM cycle when the INITiate command is sent. After the measurement, the Model 6482 returns to the idle state.

Operation summary

The trigger model is designed to make the various source-measure applications more versatile. Typically, it allows you to perform a specified number of measurements at various source levels.

For example, assume you want to perform three measurements each at two different channel 1 voltage bias source levels (1 V and 2 V). To do this, set the arm count to two (ARM:COUNt 2), the trigger count to three (TRIGger:COUNt 3), and use the list sourcing mode with the following defined list:

```
SOUR1:LIST:VOLT 1, 1, 1, 2, 2, 2
```

On the first pass through the trigger model, three measurements will be performed at the 1 V source level. On the second pass, three measurements will be performed at the 2 V source level. After the last measurement, the Model 6482 returns to the idle state. Note that the product of the arm count (finite value) and trigger count determines the number of measurements that are performed. In this example, six measurements are performed (2×3).

For details on the list source mode, see Section 16, "SOURce[1] and SOURce2."

Remote trigger commands

Table 8-1 summarizes remote trigger commands. These commands are covered in more detail in Section 16, "Trigger subsystem," except for *TRG, a common command covered in Section 14.

Table 8-1
Remote trigger command

Command	Description
`:INITiate`	Take Model 6482 out of the idle state.
`:ABORt`	Abort operation, return to the idle state.
`:ARM:COUNt <n>`	Set arm count (n = Count).
`:ARM:SOURce <name>`	Specify arm control source. Name = IMMediate, TLINk, TIMer, MANual, BUS, NSTest, PSTest, or BSTest.
`:ARM:TIMer <n>`	Set arm layer timer interval (n = Interval).
`:ARM:DIRection <name>`	Control arm bypass. Name = SOURce or ACCeptor.
`:ARM:ILINe <NRf>`	Select arm layer input line (NRf = input line number).
`:ARM:OLINe <NRf>`	Select arm layer output line (NRf = output line number).
`:ARM:OUTPut <event list>`	Select arm layer output events. Event list = TRIGger or NONE.
`:TRIGger:CLEar`	Clear any pending input triggers immediately.
`:TRIGger:COUNt <n>`	Set trigger count (n = count).
`:TRIGger:DELay <n>`	Set trigger delay (n = delay).
`:TRIGger:SOURce <name>`	Specify trigger control source. Name = IMMediate or TLINk.
`:TRIGger:DIRection <name>`	Control trigger bypass. Name = SOURce or ACCeptor.
`:TRIGger:ILINe <NRf>`	Select trigger layer input line (NRf = input line).
`:TRIGger:OLINe <NRf>`	Select trigger layer output line (NRf = output line).
`:TRIGger:INPut <event list>`	Select trigger input layer events. Event list = SOURce, DELay, SENSe, or NONE.
`:TRIGger:OUTPut <event list>`	Select trigger layer output events. Event list = SOURce, DELay, SENSe, or NONE.
`*TRG`	Trigger Model 6482 (if BUS source is selected).

Remote trigger example

Table 8-2 summarizes the command sequence for basic trigger operation. These commands set up Model 6482 triggering as follows:

- Arm layer source: Bus
- Arm layer count: 2
- Trigger layer delay: 0.1 s
- Trigger layer count: 10
- Trigger layer output events: Source and sense
- Trigger layer trigger link output line: 1

After the instrument is set up, :INIT is sent to take the instrument out of the idle state. *TRG is sent to trigger the instrument, after which it cycles 10 times through the trigger layer. A second *TRG is required to trigger the instrument the second time, and it then completes the second cycle through the trigger layer.

NOTE   *You must allow sufficient time between the first and second *TRG commands, or the second trigger will be ignored.*

Table 8-2
Remote triggering example

Command	Description
`*RST`	Restore GPIB defaults.
`:SOUR1:VOLT 10`	Source 10 V on channel 1.
`:ARM:SOUR BUS`	Select bus arm layer source (*TRG command).
`:ARM:COUN 2`	Arm layer count = 2.
`:TRIG:DEL 0.1`	0.1 s trigger layer delay.
`:TRIG:COUN 10`	Trigger layer count = 10.
`:TRIG:OUTP SOUR,SENS`	Source, sense output trigger events.
`:TRIG:OLIN 1`	Trigger output line = #1.
`:OUTP1 ON`	Turn on channel 1 output.
`:INIT`	Take instrument out of the idle state.
`*TRG`	Trigger first sequence.
`*TRG`	Trigger second sequence.
`:OUTP1 OFF`	Turn off output 1.
`:FETC?`	Request readings.

Measurement Concepts

In this section:

Source-delay-measure cycle

Overview

Model 6482 voltage bias and measurements operation for each channel can consist of a series of source-delay-measure (SDM) cycles (Figure 9-1). During each SDM cycle, the following occurs:

1. Set the voltage bias source output level.
2. Wait for the programmed delay period.
3. Make the measurement.

Figure 9-1
Source-delay-measure (SDM) cycle
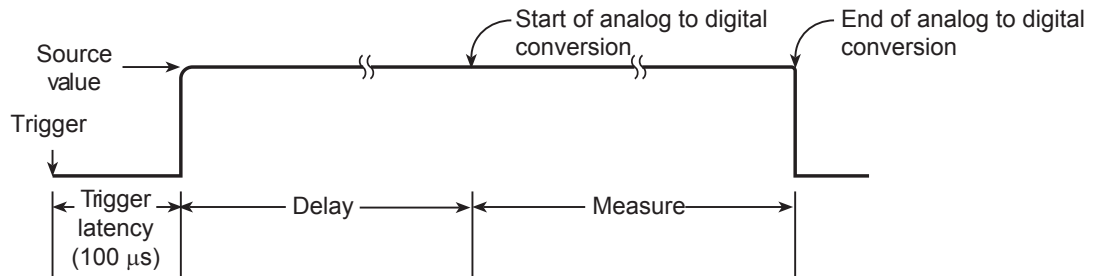
Triggering

Figure 9-2 shows how the SDM cycle fits into the trigger model. See Section 8 for complete details on the trigger model. When the source is turned on (triggered), an approximate 100 μs trigger latency occurs before the programmed source level is output. If the source output stays on, trigger latency will not be included in subsequent SDM cycles. Trigger latency only occurs when the output makes the transition from off to on.

Figure 9-2
Simplified trigger model
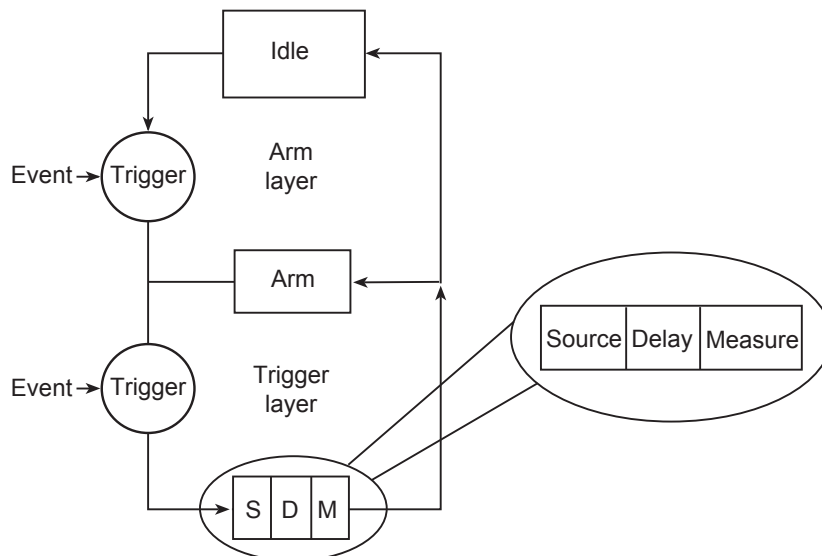
> **NOTE**   *See Section 8 for trigger model details.*

Delay phase

The delay phase of the source-delay-measure (SDM) cycle allows the source to settle before the measurement is performed. The delay period depends on how the source delay is configured. The source delay can be set manually from 0000.0000 s to 9999.9980 s. If using autodelay, the delay is set automatically; see Section 3 for details.

The manually set delay (up to 9999.998 sec) is available to compensate for longer settling required by external circuitry. The more capacitance measured at the output, the more settling time is required for the voltage bias source. The actual delay period needed can be calculated or determined by experimentation with different values. The slew rate of the output circuit is constant at about 3.07 ms/V. Therefore, the minimum source delay must be 3.07 ms $\times$ voltage step size.

> **NOTE**   *The delay period is the same for both channels. If you change the delay on one channel, the delay period on the other channel will change to the same value.*

Measurement time

The measure time depends on the selected measurement speed. For example, if speed is set at 0.01 PLC (power line cycles), the measure time would be 167 µsec for 60 Hz operation (0.01/60).

Sweep waveforms

There are three basic sweep types to select from: Linear staircase, logarithmic staircase, and custom, as shown in Figure 9-3.

Staircase sweeps

The linear staircase sweep goes from the start level to the stop level in equal linear steps. The logarithmic staircase sweep is similar to the linear staircase sweep, except it is done on a logarithmic scale with a specified number of steps per decade.

Custom sweep

The custom sweep lets you construct your own sweep by specifying the number of measure points and the source level at each point. See Section 7 for more details on sweep operation.

SDM cycle during sweeps

A source-delay-measure (SDM) cycle is performed on each step (or point) of the sweep; one measurement will be performed at each step (level). The time spent at each step (level) depends on how the SDM cycle is configured (in other words, the source delay and measure speed configuration) and the trigger delay (if used).

Figure 9-3
Basic sweep waveform types

Linear staircase sweep

Logarithmic scale
shown for staircase steps.

Logarithmic staircase sweep
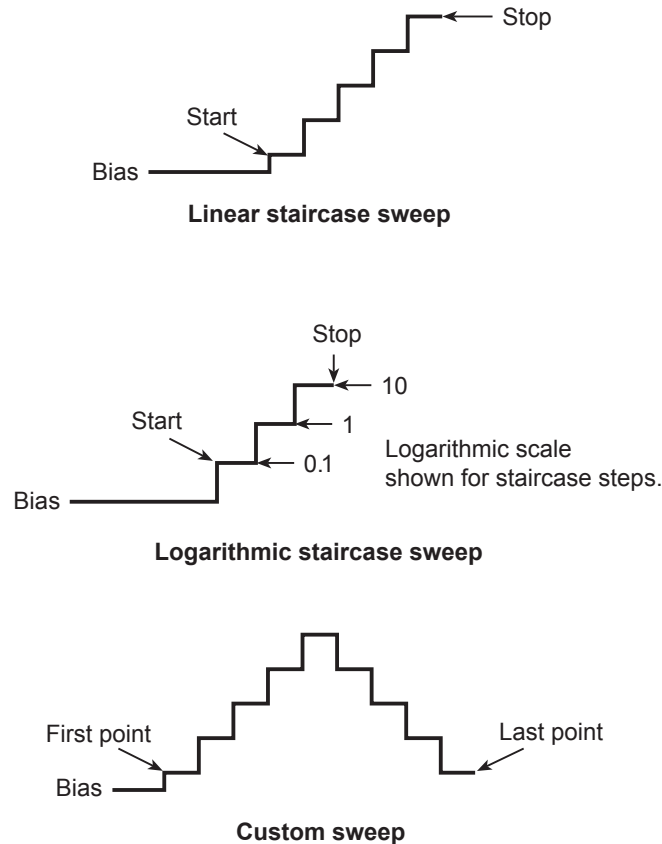
Custom sweep

Typical sweep applications

Typical applications for staircase sweeps include I-V curves. The custom sweep can be used to configure a pulse sweep with a 50 percent duty cycle. For example, a 10 V pulse sweep can be configured by programming the odd numbered points for 10 V and the even numbered points for 0 V. When the sweep is run, the output will alternate between 10 V and 0 V.

Sweep data storage

For a sweep that has a finite sweep count, the data will be automatically stored in the buffer. This data can be accessed from the front panel or sent to a computer (remote operation) for evaluation (plotting). Statistical information on readings stored in the buffer are also available from the front panel or over a remote interface.

Bias source operating boundaries

Limit lines

Figure 9-4 shows the operating boundaries (limit lines) for the voltage bias sources in the first quadrant (both voltage and current positive). Operation in the third quadrant (both voltage and current negative) is similar. Note that each voltage bias source can output a maximum of ±30 V at ±20 mA. Although the voltage value can be set over a range of ±30 V, the current compliance is fixed at 20 mA.

Figure 9-4
Bias source limit lines
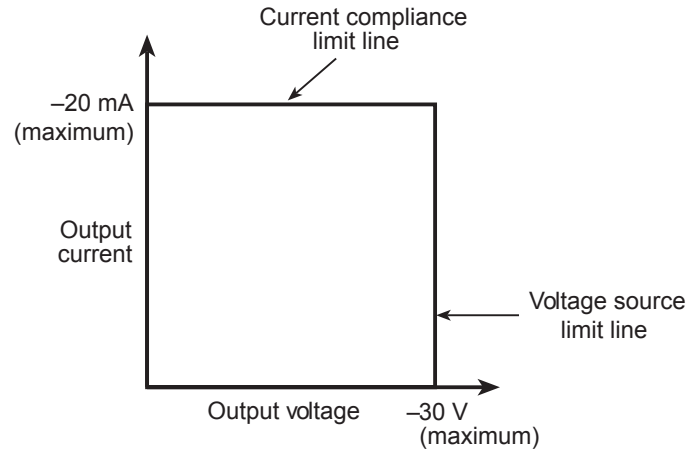
Loading effects

The specific location within the boundaries each bias source operates depends on the resistance of the load (also called the device under test or DUT) that is connected to the output. Figure 9-5 shows operation examples for resistive loads that are 1 kΩ and 400 Ω, respectively. For these examples, one bias source (of the Model 6482) is programmed to source 10 V with a fixed current limit of 20 mA.

In Figure 9-5A, the Model 6482 is sourcing 10 V into the 1 kΩ load, and subsequently sources 10 mA. As shown, the load line for 1 kΩ intersects the 10 V voltage source line at 10 mA.

Figure 9-5B shows what happens if the resistance of the load is decreased to 400 Ω . The DUT load line for 400 Ω intersects the 20 mA current compliance limit line, placing the Model 6482 in compliance. In compliance, the Model 6482 will not be able to source its programmed voltage (10 V). For the 400 Ω DUT, the instrument will output only 8 V (at the fixed 20 mA limit).

Notice that as resistance increases, the slope of the DUT load line decreases. As resistance approaches infinity (open output), the Model 6482 will source virtually 10 V at 0 mA. Conversely, as resistance decreases, the slope of the DUT load line increases. At zero resistance (shorted output), the Model 6482 will source virtually 0 V at 20 mA.

Regardless of the load, current will never exceed the fixed compliance of 20 mA.

Figure 9-5
Loading effects

$$I = V / R$$
$$= 10 \text{ V}/1 \text{ k}\Omega$$
$$= 10 \text{ mA}$$
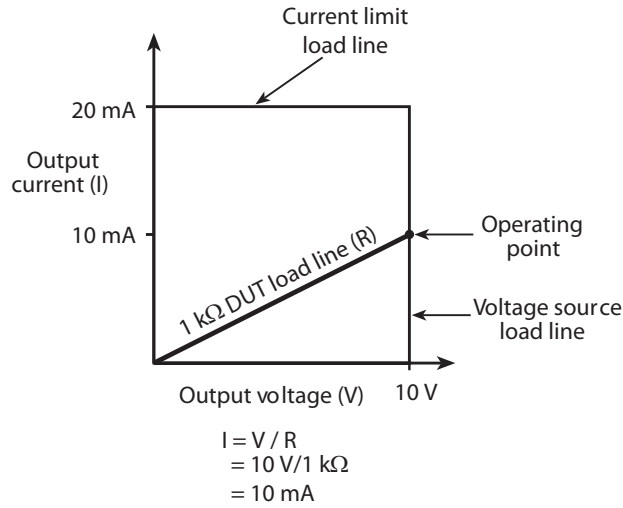
A. Normal bias source operation

$$V_S = I_M \cdot R$$
$$= (20 \text{ mA}) (400 \ \Omega)$$
$$= 8 \text{ V}$$
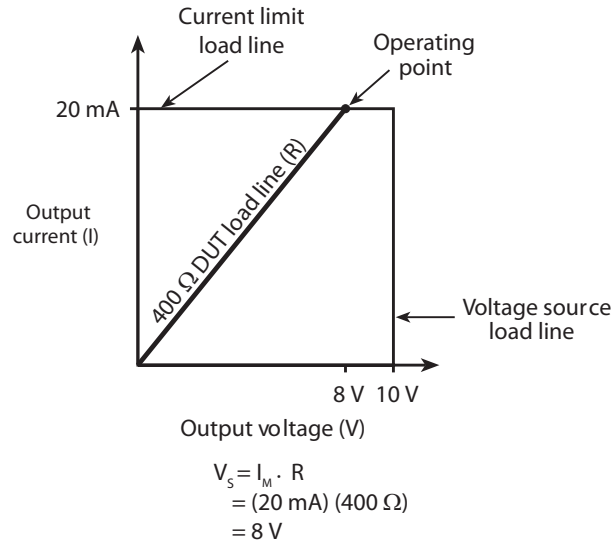
B. Bias source in compliance

Data flow

Data flow for front-panel operation is summarized by the block diagrams provided in Figure 9-6, and Figure 9-7 shows data flow through the various CALC blocks. Note that if REL is enabled, the result of the rel operation is sent to the other blocks.

NOTE   See *Appendix B* for remote operation data flow information that describes the CALC blocks in detail.

Basic readings

With data store and limit tests disabled (Figure 9-6A), the Model 6482 displays the selected measurement function readings (MSR1, MSR2, RATIO, or DELTA); no data storage or limit test is performed when those two functions are disabled.

Data storage enabled

With data store enabled (Figure 9-6B), all readings are stored in the data store buffer as processed regardless of the selected measurement function before or during storage. The type of recalled reading, or statistic, depends on the selected measurement function during recall (MSR1, MSR2, RATIO, or DELTA).

Limit test enabled

With a limit test enabled (Figure 9-6C), the selected limit feed (MSR1, MSR2, RATIO, or DELTA) is used for Limit 3 through Limit 6 tests. Pass or fail (P or F) information is displayed on the front panel with the readings. If data store is performed with a limit test already enabled, pass or fail status information is also displayed with the readings during data recall.

Figure 9-6
Data flow front panel
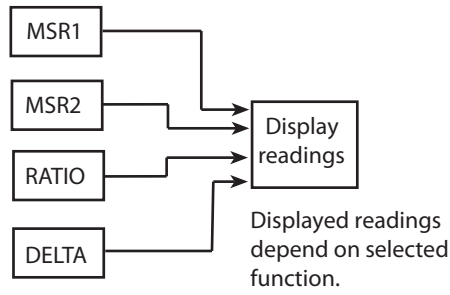
A. Data store and limit tests disabled
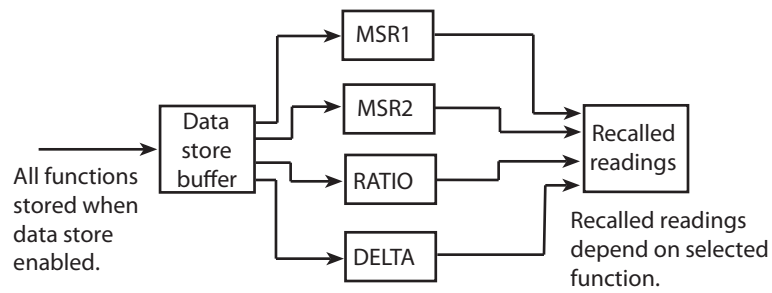
B. Data store enabled
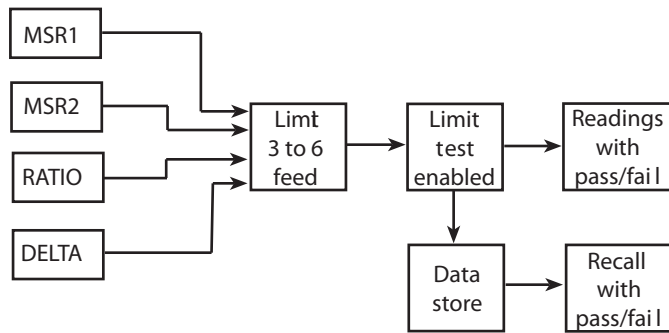
C. Limit tests and data store enabled

Figure 9-7
CALC block data flow
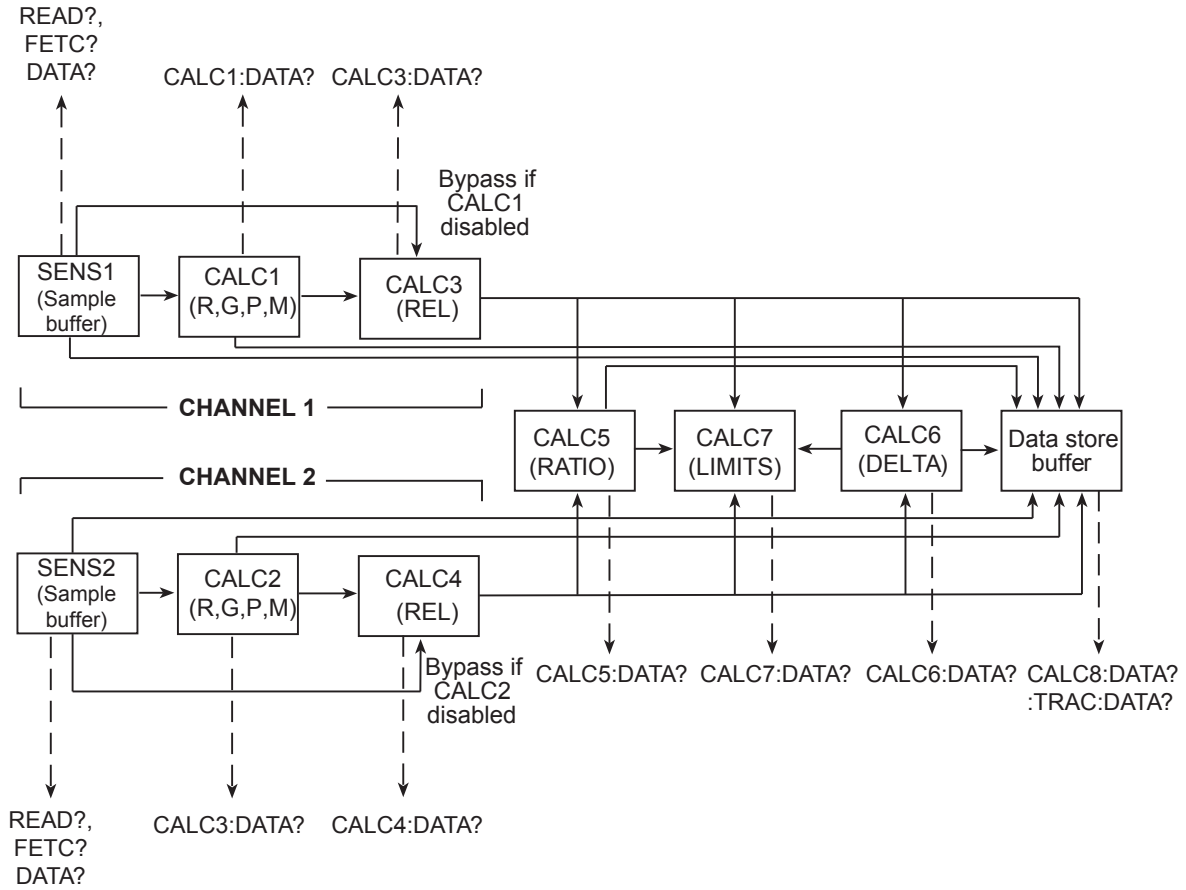
This page left blank intentionally.

In this section:

Types of limits

There are six limit tests that can be performed on a DUT.

- Limit 1: channel 1 hardware limit (compliance) test.
- Limit 2: channel 2 hardware limit (compliance) test.
- Limits 3 to 6: software limits for channel 1, channel 2, ratio, and delta.

A test is only performed if it is enabled; you can perform one, two, or all six tests. The tests are always performed in the order shown in Figure 10-1.

Figure 10-1
Limit tests
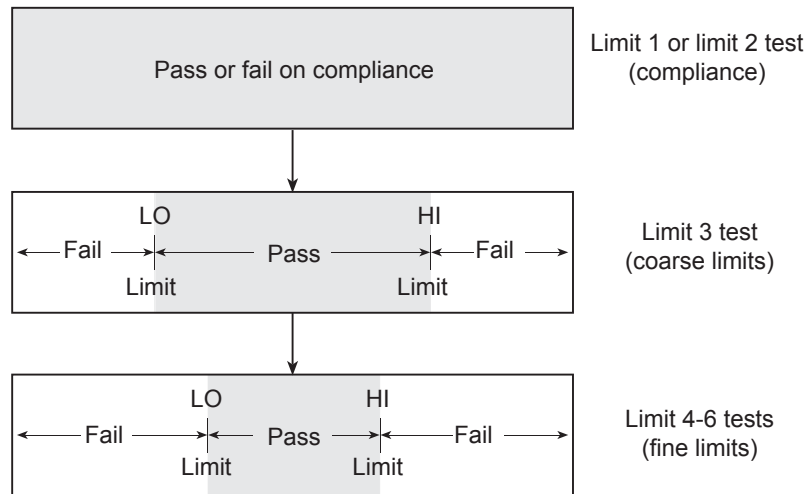
Pass or fail information

Pass or fail information for limit tests can be obtained as follows:

- A PASS or FAIL indication on the front panel display.
- With the :CALCulate7:LIMit<n>:FAIL? query over a remote interface, where <n> is the limit test number (Section 16, "CALCulate7").
- By reading various status bits (Section 13, "Status Structure," and Section 16, "FORMat subsystem").
- By noting a P or F preceding buffer location numbers (Section 6, "Buffer location number").

Data flow

All limit tests are part of the CALC7 data block. See Appendix B for an overview on how limit testing fits into the overall data flow through the Model 6482.

Limit test feeds

You can use the following feeds (data sources) for the software limit tests (Limit 3 to Limit 6): MSR1, MSR2, RATIO, and DELTA. See "Configuring and performing limit tests" on page 10-5 for more information..

Limit 1 and 2 tests (compliance)

The limit 1 and limit 2 tests are hardware (H/W) tests that check the voltage source compliance states of the Model 6482. Limit 1 uses the channel 1 voltage source current compliance current value (20 mA) as the limit, while Limit 2 uses the channel 2 voltage source compliance current as the limit. At or above the 20 mA current limit, the instrument is in compliance. Below the current limit, the instrument is not in compliance.

Limit 3 to 6 tests

The limit 3 through limit 6 tests are software (S/W) tests that are used to determine if a DUT is within specified high and low limits. Typically, the Limit 3 test is used to test for coarse tolerance limits, and the Limit 4 through Limit 6 tests are used for progressively finer tolerance limits.

For example, assume you want to sort diodes into three groups: 10%, 15%, and >20% current tolerance. To do this, configure the Limit 3 test for 20% HI and LO limits and Limit 4 for 15% HI and LO limits. If Limit 3 fails, the :CALCulate7:LIMit3:FAIL? command will return a 0 (limit test passed). If Limit 3 passes, the Limit 4 test is run. If Limit 4 fails, the :CALCulate7:LIMit4:FAIL? command will return a 1 (limit test failed). If Limit 4 passes, the :CALCulate7:LIMit4:FAIL? command return a 0 (limit test passed).

Although Limit 1 and Limit 2 (discussed above) are dedicated to channels 1 and 2 respectively, the Limit 3 through Limit 6 tests can be used with channel 1, channel 2, RATIO, or DELTA measurements by selecting the corresponding data feed. See "Configuring limit tests," page Section 10-5, as well as Appendix B, "Data Flow."

Limit tests

If the limit 1 or limit 2 test fails, the FAIL message is displayed and the testing process for that DUT (or DUT element) is terminated. A pass condition allows the testing process to proceed to the next enabled limit test.

Each enabled software test (limit 3 through 6) is performed until a test passes. When a test passes, the PASS message is displayed and the testing process for that DUT is terminated.

Operation overview

Limits operation is detailed by the flow chart in Figure 10-2. A test is only performed if it is enabled; if a test is disabled, operation proceeds to the next test. The following assumes the first three limit tests are enabled.

With the limit tests properly configured, turn the Model 6482 output on, and then press the **LIMIT** key. The testing process will start when an SOT signal SOT (start-of-test) strobe pulse is received by the Model 6482. Note that if the trigger model is not set to accept an SOT signal, testing will start when the **LIMIT** key is pressed. Pressing the **LIMIT** key a second time terminates the testing process.

As shown in the flow chart, limit tests are performed after a measurement conversion.

Figure 10-2
Limit testing model

Start — Turn the output ON and press the **LIMIT** key

Wait for SOT pulse (if enabled)*

*   If the trigger model is not set to accept an SOT signal, testing will start when the **LIMIT** key is pressed.

Perform source-measure action

Perform limit 1/2 test? — Yes → Pass? — No → Display "FAIL" → :CALCulate7:LIMit$x$:FAIL? can be read as 1 (limit test failed). x = 1 or 2

Pass? — Yes → Limits 3-6 Disabled? — Yes → Display "PASS" → :CALCulate7:LIMit$x$:FAIL? can be read as 0 (limit test passed). x = 3 to 6

Limits 3-6 Disabled? — No

Perform limit 3 test? — Yes → Pass? — Yes → Display "PASS" → :CALCulate7:LIMit3:FAIL? can be read as 0 (limit test passed).

Pass? — No

Perform limit 4-6 tests? — Yes → Pass? — Yes → Display "PASS" → :CALCulate7:LIMit$x$:FAIL? can be read as 0 (limit test passed). x = 4 to 6

Pass? — No

Display "FAIL"

Test Another Device? — Yes / No

Stop — Press the **LIMIT** key

Pass condition

After the tests pass, the PASS message is displayed. Note that the pass condition can also be determined with the :CALC7:LIM<n>FAIL? query over a remote interface.

If the Model 6482 is configured to test another DUT package, operation loops back to the top of the flow chart and waits for the start-of-test (SOT) pulse.

Fail condition

When a failure occurs, the FAIL message is displayed and can also be read over a remote interface with :CALC7:LIM<n>FAIL? command.

If configured to test another DUT package, operation loops to the top of the flow chart and waits for the start-of-test (SOT) pulse.

Output enable connector

See Section 11, "Output enable," for more information.

SOT line

The input line (SOT) of the output enable connector is used to control the start of the testing process. When ↓STEST is the selected arm event of the trigger model, the testing process will start when the SOT line is pulsed low. When ↑STEST is the selected arm event, the testing process will start when the SOT line is pulsed high. When ↑↓STEST is the selected arm event, the testing process will start when the SOT line is pulsed either high or low. With the IMMEDIATE arm event selected, the testing process will start as soon as the LIMIT key is pressed (assuming the output is ON). See Section 8 for details on trigger model configuration and Section 11 for details on the output enable connector.

Output enable (/OE) line

NOTE   *The forward slash (as shown /OE) designates that the pin is active low (in other words, when connected to ground).*

With proper use of output enable, power is removed from the DUT when the lid is opened. See Section 11, "Output enable," for operation details.

Configuring and performing limit tests

Configuring limit tests

Press the **CONFIG** key and then the **LIMIT** key to display the CONFIG LIMITS MENU. The limits configuration menu is structured shown below and in Figure 10-3. The limits configuration menu is structured as follows. Note that bullets indicate the primary items of the limit menu and dashes indicate the options of each menu item. Refer to Section 1, "Rules to navigate menus," to configure the limit tests.

- **H/W-LIMITS**: Use this menu item to control and set the fail mode for the Limit 1 (channel 1) and Limit 2 (channel 2) compliance tests:
  - **CONTROL**: Use to ENABLE or DISABLE the test.
  - **FAIL-MODE**: Use to select the fail mode for Limit 1 and Limit 2 tests. With IN selected, the test will fail when the Model 6482 is in compliance. With OUT selected, the test will fail when not in compliance.

- **S/W-LIMITS**: Use this menu item to control, set limits for, and define output bit patterns for Limit 3 to Limit 6 tests:
  - **CONTROL**: Use to ENABLE or DISABLE the test.
  - **LOLIM**: Use to set the low limit.
  - **HILIM**: Use to set the high limit.
  - **FEED**: Use to select the data source for the Limit 3 through Limit 6 tests. Specify MSR1 or MSR2 (channel 1 or channel 2 measurement function), RATIO, or DELTA.

Figure 10-3
Limits configuration menu tree
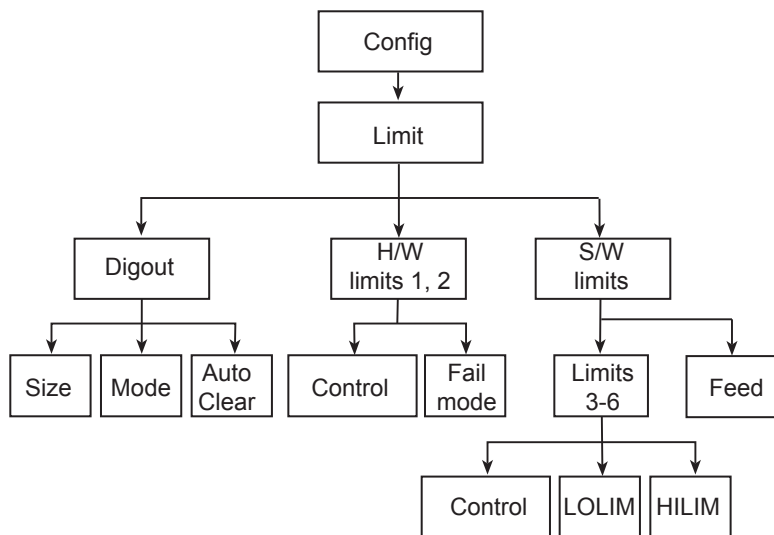
Performing limit tests

Perform the following steps to run limit tests:

Step 1. Configure test system.

As previously explained in Section 2, your test system could be as simple as connecting one or two DUTs to the Model 6482. Adding a scanner to the test system allows you to test multiple devices.

Step 2. Configure bias source and measure functions.

Configure the Model 6482 operation as follows:

1. Select the source channel by pressing the **SRC1** or **SRC2** key.
2. Set the source level value.
3. Press the **MSR1** or **MSR2** key to select the measurement channel, then choose the measurement range.

Refer to Section 3, "Basic Operation," for more information.

Step 3. Configure limit tests.

Select and configure the following limit tests parameters as explained in Configuring limit tests:

- Set the H/W-LIMITS and S/W-LIMITS parameters.

- Remember that Limit 1 and Limit 2 are the compliance limits for channel 1 and channel 2 respectively, while the data source for Limit 3 through Limit 6 depends on the selected FEED (MSR1, MSR2, RATIO, or DELTA).

Step 4. Turn output on.

Press the **ON/OFF** key to turn the output on (blue OUTPUT indicator turns on). The Model 6482 will output the programmed bias voltage level.

Step 5. Start testing process.

To enable the limit tests, press the **LIMIT** key. If the /SOT line of the OUTPUT ENABLE connector is set to receive a low-going pulse a /SOT pulse, the testing process will not start until /SOT line receives a low-going pulse. Otherwise the testing process will start when **LIMIT** key is pressed.

NOTE   *The SOT line may be configured for a high-going pulse, or configured for either a low or high-going pulse. See "Event detection" on page 8-3 for more information..*

Step 6. Stop testing process.

The testing process can be terminated at any time by again pressing the **LIMIT** key.

Remote limit testing

Limit commands

Table 10-1 summarizes remote commands to control limit testing. Note that LIMitX refers to identical software limits LIMit3 through LIMit6. See Section 16, "CALCulate7" for more details on these commands.

Table 10-1
Limit commands

Command	Description
:CALCulate7:LIMit[1]:COMPliance:FAIL <name>	Set Limit 1 fail condition. Name = IN (fail into compliance) or OUT (fail out of compliance).[1]
:CALCulate7:LIMit[1]:STATe <state>	Enable or disable Limit 1 test (state = ON or OFF).
:CALCulate7:LIMit[1]:FAIL?	Query Limit 1 test result (0 = pass, 1 = fail).
:CALCulate7:LIMit2:COMPliance:FAIL <name>	Set Limit 2 fail condition. Name = IN (fail into compliance) or OUT (fail out of compliance).
:CALCulate7:LIMit2:STATe <state>	Enable or disable Limit 2 test (state = ON or OFF).
:CALCulate7:LIMit2:FAIL?	Query Limit 2 test result (0 = pass, 1 = fail).
:CALCulate7:FEED <name>	Select Limit 3-6 tests input path (name = CALC3, CALC4, CALC5, or CALC6).[2]
:CALCulate7:LIMitX:LOWer <n>	Specify lower Limit X. [3]
:CALCulate7:LIMitX:UPPer <n>	Specify upper Limit X.
:CALCulate7:LIMitX:STATe <state>	Enable or disable Limit X test (state = ON or OFF).
:CALCulate7:LIMitX:FAIL?	Query Limit X test result (0 = pass, 1 = fail).
:CALCulate7:CLIMits:CLEar	Clear test results, reset I/O port.
:CALCulate7:CLIMits:CLEar:AUTO <state>	Enable or disable auto-clear (state = ON or OFF).

[1] LIMit[1] dedicated to channel 1; LIMit2 dedicated to channel 2 (hardware limits).
[2] CALC3 = channel 1 measurement; CALC4 = channel 2 measurement; CALC5 = RATIO; CALC6 = DELTA.
[3] LIMitX = LIMit3, LIMit4, LIMit5, and LIMit6 (software limits).

Limit test programming example

Diode current test is an example that readily lends itself to pass or fail analysis. This test verifies the forward currents at various voltages. The test is performed by sourcing a specified voltage level and then measuring the current. The current is then compared with one set of limits to determine if the diode passes or fails and should be discarded. A marginal diode, determined by a second set of narrower limits, can be sent to Q/A for further analysis.

Test parameters for this test include:

- Source and measure channel: channel 1
- Source voltage: 1 V
- Source delay: 100 ms
- Limit 3 upper value: 10 mA
- Limit 3 lower value: 4 mA
- Limit 4 upper value: 8 mA
- Limit 4 lower value: 6 mA

Table 10-2 summarizes the basic SCPI command sequence for performing a limit test for the diode current test.

NOTE   *See Appendix F for a complete program listing that includes additional programming steps necessary to test the values returned by the :CALC7:LIM3:FAIL? and :CALC7:LIM4:FAIL? queries.*

Table 10-2
Limits test programming example

Command	Description
`*RST`	Restore GPIB default conditions.
`:SENS1:CURR:RANG:AUTO ON`	Turn on channel 1 measure autorange.
`:SOUR1:VOLT:TRIG 1`	Output 1 V on channel 1 when triggered.
`:SOUR1:DEL 0.1`	100 ms source delay.
`:CALC7:FEED CALC3`	Use CALC3 (Channel 1) data for limits comparison.
`:CALC7:LIM3:UPP 10e-3`	Limit 3 upper value = 10 mA.
`:CALC7:LIM3:LOW 4e-3`	Limit 3 lower value = 4 mA.
`:CALC7:LIM4:UPP 8e-3`	Limit 4 upper value = 8 mA
`:CALC7:LIM4:LOW 6e-3`	Limit 4 lower value = 6 mA.
`:CALC7:LIM1:STAT 0`	Turn off Limit 1 test.
`:CALC7:LIM3:STAT 1`	Turn on Limit 3 test.
`:CALC7:LIM4:STAT 1`	Turn on Limit 4 test.
`:OUTP1 ON`	Turn on channel 1 source output.
`:INIT`	Trigger reading and limits test.
`:OUTP1 OFF`	Turn off output.
`:CALC7:LIM3:FAIL?`	Query Limit 3 test results (1 = discard diode).
`:CALC7:LIM4:FAIL?`	Query Limit 4 test results (1 = send diode to QA).

Output Enable and Output Configuration

In this section:

Output enable

The OUTPUT ENABLE connector provides the output enable line for use with a test fixture switch. When properly used, the voltage source outputs of the Model 6482 will turn OFF when the lid of the test fixture is opened.

> *WARNING*   ***To prevent electric shock, test connections must be configured so that the user cannot come in contact with conductors or any device under test (DUT) that is in contact with the conductors. Safe installation requires proper shields, barriers, and grounding to prevent contact with conductors. Operator protection and safety are the responsibility of the person installing the product.***

The output enable connector is located on the rear panel and is shown in Figure 11-1. Note that a standard male DB-9 connector is used for the output enable connector. A custom cable using a standard female DB-9 connector is required for connection to the output enable connector.

Figure 11-1
Output enable connector

Model 6482

Pin	Description
1-4	Not connected
5	Ground
6	Input (SOT)
7	+5 V DC
8	Output enable (/OE)
9	Ground
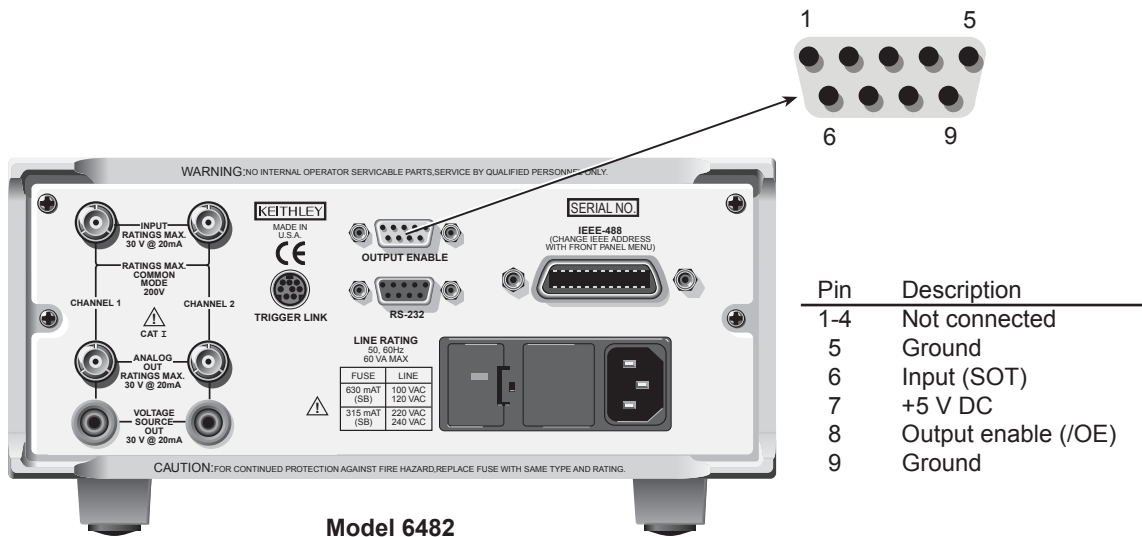
Ground

Pin 5 or pin 9 can be used as a ground for the output enable (see Figure 11-2).

Input (SOT) line

The start of test (SOT) input line on pin 6 could be used to start limit testing. Dependent on the arm event selected, providing the corresponding SOT pulse will move the trigger model from the arm layer into the trigger layer and start the test:

- With the ↓STEST arm event selected, provide a low pulse (SOT low) to move from the arm layer into the trigger layer and start the test.
- In a similar manner, with the ↑STEST arm event selected, provide a high pulse (SOT high) to start the test.
- With the ↑↓STEST arm event selected, provide a low or a high pulse (SOT low or SOT high) to start the test.

For front-panel operation of the trigger model, see "Event detection" on page 8-3; for remote operation, see "Event detection" on page 8-15.

+5 V DC

The OUTPUT ENABLE connector provides a +5 V DC output on pin 7 that can be used to drive external logic circuitry. Maximum current output for this line is 300 mA. This line is protected by a self-resetting fuse (one hour recovery time).

Output enable (/OE)

NOTE   *The forward slash (as shown /OE) designates that the pin is active low (in other words, when connected to ground).*

When output enable is activated (see "Front-panel output configuration," page Section 11-4), the output of the Model 6482 cannot be turned on unless the output enable line is pulled low through a switch to ground as shown in Figure 11-2A. If the lid of the test fixture opens (Figure 11-2B), the switch opens, and the output enable line goes high turning the output of the Model 6482 OFF (high impedance). The output can only be turned back on by first closing the lid of the test fixture and then pressing the OUTPUT ON/OFF key.

Figure 11-2
Using test fixture output
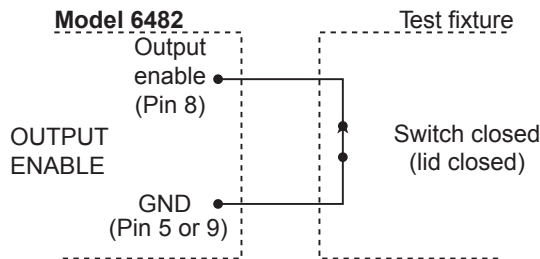
A. Model 6482 output can be turned on.
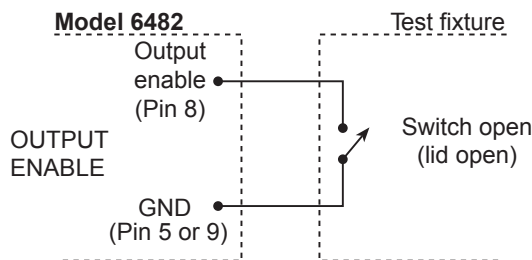
B. Model 6482 ouput turns off.

Front-panel output configuration

The output is configured from the CONFIGURE OUTPUT menu and is structured as described in the following topics. Note that bullets indicate the primary items of the sweep menu, while dashes indicate options. Use Section 1, "Rules to navigate menus," to check and change operate options.

Configure OUTPUT menu

Press CONFIG and then ON/OFF OUTPUT to display the menu, which is also shown in Figure 11-3.

- **AUTO-OFF:** Use to ENABLE or DISABLE automatic output off. With ENABLE and ALWAYS enabled, the OUTPUT will turn off after the measurement phase of every source, delay, and measurement (SDM) cycle. The OUTPUT turns back on at the beginning of the next SDM cycle. With ENABLE and AFTER-TRIG-COUNT, the output will remain on until the trigger count expires. When disabled, the OUTPUT stays on as long as the Model 6482 is operating within the trigger model (ARM indicator on). With the OUTPUT enabled, pressing the ON/OFF key will disable the OUTPUT and disable automatic output off.

- **ENABLE**: Use to ENABLE or DISABLE the output enable line for use with a test fixture. See "Output enable," page Section 11-2.

Figure 11-3
Output configuration menu tree
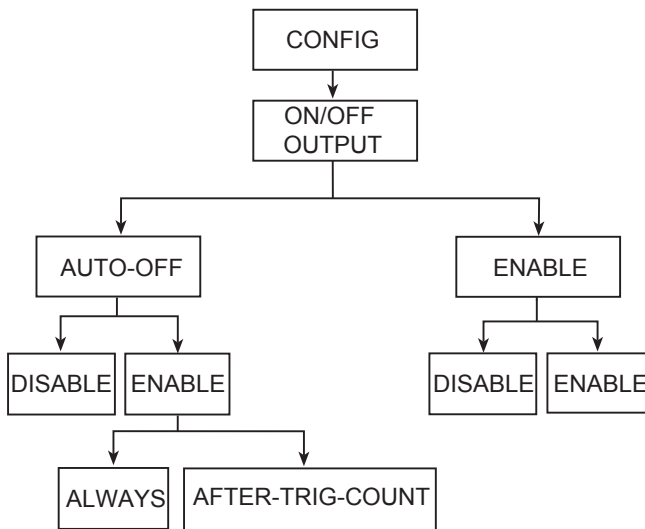
Remote output configuration

This topic summarizes the remote commands used to control the output enable connector and main output off states. A simple programming example is also provided.

Output configuration commands

Table 11-1 summarizes output configuration commands. These commands include those to enable and disable the output enable line and commands to control output off states. See Section 16, "OUTPut subsystem" and "SOURce subsystem," for more information.

NOTE   *Output enable and source clear commands are global and affect both channels simultaneously.*

Table 11-1
Output configuration commands

Command	Description
`:OUTPut[1]:ENABle <state>`	Enable or disable output enable (state = ON or OFF).
`:OUTPut[1]:ENABle:TRIPped?`	Query output enable tripped state (1 = Tripped).
`:SOURce[1]:CLEar`	Turn output source off when in idle state.
`:SOURce[1]:CLEar:AUTO <state>`	Enable or disable automatic output off. State = ON (output off after measurement) or OFF (output stays on).
`:SOURce[1]:CLEar:AUTO:MODE <name>`	Automatic clear mode. Name = ALWays (every reading; default) or TCOunt (ON when trigger layer entered; OFF when leaving trigger layer).
`:SOURce2:CLEar`	Turn output source off when in idle state.
`:SOURce2:CLEar:AUTO <state>`	Enable or disable automatic output off. State = ON (output off after measurement) or OFF (output stays on).
`:SOURce2:CLEar:AUTO:MODE <name>`	Automatic clear mode. Name = ALWays (every reading; default) or TCOunt (ON when trigger layer entered; OFF when leaving trigger layer).

Output configuration programming example

Table 11-2 lists the command sequence for output configuration. These commands set up the Model 6482 as follows:

- Output enable: On
- Auto-off mode: On

NOTE   *Connect pins 8 and 9 of the output enable connector to each other to simulate a closed output enable switch. If you do not connect pins 8 and 9, the instrument will not turn on its output when the measurement is made.*

Table 11-2
Output configuration programming example

Command	Description
`*RST`	Restore GPIB defaults.
`:FORM:ELEM CURR1`	Select channel 1 reading data.
`:SOUR1:VOLT 10`	Output 10 V on channel 1.
`:OUTP:ENAB ON`	Activate output enable.*
`:SOUR1:CLE:AUTO ON`	Enable automatic off (auto-off) mode.
`:READ?`	Trigger and acquire readings.

*Connect pins 8 and 9 of the output enable connector to simulate closed output enable switch.

This page left blank intentionally.

In this section:

Remote versus local operation

This topic summarizes remote operation enhancements and local-to-remote and remote-to-local transitions.

Local-to-remote transition

When changing from local-to-remote operation, the following takes place:

- The Model 6482 stops taking readings and is placed into the idle layer of the trigger model.
- All menus are exited.
- All pending front-panel commands are aborted.
- The reading buffer is cleared; FETCh?, CALC1:DATA?, and CALC2:DATA? will not return any data until the Model 6482 takes readings while in remote operation.
- All other settings remain unaffected, including TRACe buffer storage.
- Sources are placed in the FIXed mode.

Remote-to-local transition

When changing from remote-to-local operation, the following takes place:

- The Model 6482 stops taking readings and is placed into the idle layer of the trigger model.
- All user-defined displays are disabled.
- The display is re-enabled if it was previously turned off.
- Readings are continuously taken.
- Sources are placed in the FIXed mode.

Selecting an interface

The Model 6482 supports two built-in remote interfaces:

- GPIB (general purpose interface bus)
- RS-232 interface

You can use only one interface at a time. The factory interface selection is the GPIB bus. You can select the interface only from the front panel. The interface selection is stored in nonvolatile memory; it does not change when power has been off or after a remote interface reset.

The GPIB bus is the IEEE-488 interface. You must select a unique address for the Model 6482. The address is displayed when the instrument is turned on. At the factory, the address is set to 25. The IEEE-488 address can only be changed from the front panel while the IEEE-488 bus is enabled.

The RS-232 interface is a serial interface. Programmable attributes of the RS-232 interface include the following (factory default settings are shown in parentheses):

- Baud rate (9600)
- Data bits (8)
- Parity (none)
- Terminator (CR)
- Flow control (none)

RS-232 settings for baud rate, parity, and data bits are accessible only from the front panel while RS-232 is enabled.

An interface is selected and configured from the COMMUNICATIONS option of the main menu. See Section 1, "Main menu." For details on the programmable aspects of the interfaces, see "Primary address," page Section 12-5 and "RS-232 interface operation," page Section 12-13.

NOTE   *When changing interface selections, the Model 6482 performs a power-on reset. To check or change options of the selected interface, you must re-enter the menu structure.*

GPIB operation

This section contains information about GPIB standards, bus connections, and primary address selection.

GPIB standards

The GPIB is the IEEE-488 instrumentation data bus with hardware and programming standards originally adopted by the IEEE (Institute of Electrical and Electronic Engineers) in 1975. The Model 6482 supports IEEE Std 488.2. The Model 6482 also supports SCPI, a command language protocol. This defines a standard set of commands to control every programmable aspect of an instrument.

GPIB connections

To connect the Model 6482 to the GPIB bus, use a cable equipped with standard IEEE-488 connectors, as shown in Figure 12-1.

Figure 12-1
IEEE-488 connector
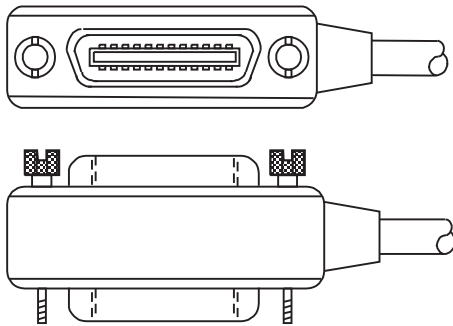
To allow many parallel connections to one instrument, stack the connectors. Two screws are located on each connector to ensure that connections remain secure. Figure 12-2 shows a typical connection scheme for a multi-instrument test system.

Figure 12-2
IEEE-488 connections
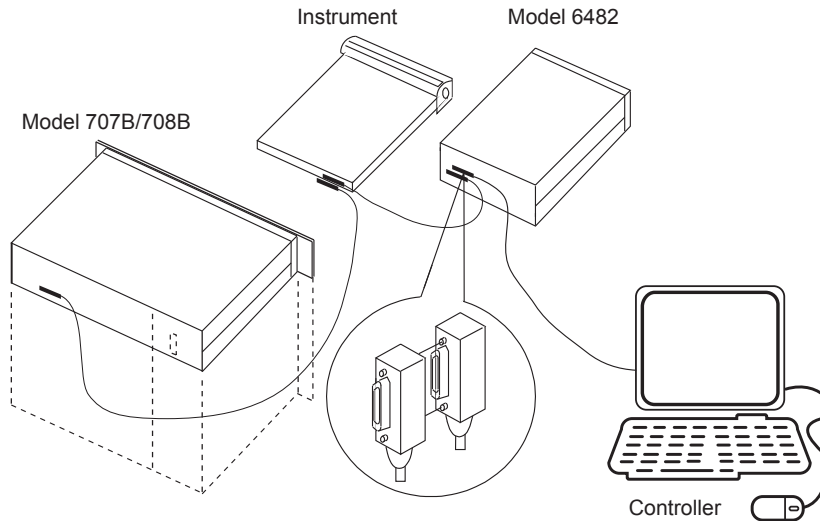
To avoid possible mechanical damage, stack no more than three connectors on any one instrument.

NOTE   *To minimize interference caused by electromagnetic radiation, use only shielded IEEE-488 cables. Available shielded cables from Keithley are Models 7007-1 and 7007-2.*

To connect the Model 6482 to the IEEE-488 bus, follow these steps:

1. Align the cable connector with the connector located on the rear panel. The connector is designed so that it will fit only one way. Figure 12-3 shows the location of the IEEE-488 connector.

2. Tighten the screws securely, making sure not to overtighten them.

Figure 12-3
IEEE-488 and RS-232 connector location

Model 6482
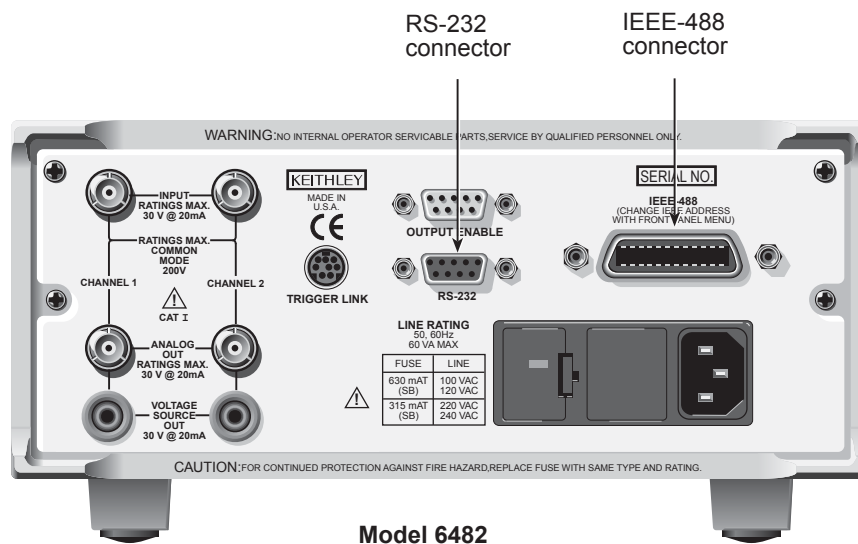
3. Connect any additional connectors from other instruments as required for your application.

4. Make sure the other end of the cable is properly connected to the controller. Most controllers are equipped with an IEEE-488 style connector, but a few may require a different type of con-

necting cable. See your controller's instruction manual for information about properly connecting to the IEEE-488 bus.

NOTE   *You can only have 15 devices connected to an IEEE-488 bus, including the controller. The maximum cable length is either 20 meters (65.6 feet) or 2 meters (6.5 feet) multiplied by the number of devices, whichever is less. Not observing these limits may cause erratic bus operation.*

Primary address

The Model 6482 ships from the factory with a GPIB primary address of 25. When the instrument powers up, it momentarily displays the primary address. You can set the address to a value from 0 to 30, but do not assign the same address to another device or to a controller that is on the same GPIB bus (controller addresses are usually 0 or 21).

The primary address can be checked or changed from the COMMUNICATIONS/GPIB option of the main menu. See Section 1, "Main menu." This menu option also allows you to select the 488.1 or SCPI protocol (Appendix E).

General bus commands

General commands are those commands, such as DCL, that have the same general meaning when used on any of the instrument. Table 12-1 lists the general bus commands.

Table 12-1
General bus commands

Command	Effect on Model 6482
REN	Goes into remote operation when next addressed to listen.
IFC	Goes into talker and listener idle states.
LLO	LOCAL key locked out.
GTL	Cancel remote; restore Model 6482 front-panel operation.
DCL	Returns all devices to known conditions.
SDC	Returns Model 6482 to known conditions.
GET	Initiates a trigger.
SPE, SPD	Serial polls the Model 6482.

REN (remote enable)

The remote enable command is sent to the Model 6482 by the controller to set up the instrument for remote operation. Generally, the instrument should be placed in the remote mode before you attempt to program it over the bus. Setting REN true does not place the instrument in the remote state. You must address the instrument to listen after setting REN true before it goes into the remote operation.

The Model 6482 must be in remote mode to use the following commands to trigger and acquire readings:

- :INITiate and then :FETCh?
- :READ?
- :MEASure?

IFC (interface clear)

The IFC command is sent by the controller to place the Model 6482 in the local, talker, and listener idle states. The instrument responds to the IFC command by canceling front-panel TALK or LSTN indicators, if the instrument was previously placed in one of these states.

Note that this command does not affect the status of the instrument. Settings, data, and event registers are not changed.

With automatic output off enabled (:SOURce1:CLEar:AUTO ON or :SOURce2:CLEar:AUTO ON), the output will remain on if operation is terminated before the output has a chance to turn off automatically.

To send the IFC command, the controller need only set the IFC line true for a minimum of 100 µs.

LLO (local lockout)

Use the LLO command to prevent local operation of the instrument. After the instrument receives an LLO command, all of its front panel controls except OUTPUT OFF are inoperative. In this state, pressing LOCAL will not restore control to the front panel. The go to local (GTL) command restores control to the front panel. Cycling power will also cancel local lockout.

GTL (go to local)

Use the GTL command to put a remote-mode instrument into local mode. The GTL command also restores front-panel key operation.

DCL (device clear)

Use the DCL command to clear the GPIB interface and return it to a known state. Note that the DCL command is not an addressed command, so all instruments equipped to implement DCL will do so simultaneously.

When the Model 6482 receives a DCL command, it clears the input buffer and output queue, cancels deferred commands, and clears any command that prevents the processing of any other device command. A DCL command does not affect instrument settings and stored data.

SDC (selective device clear)

The SDC command is an addressed command that performs essentially the same function as the DCL command. However, because each device must be individually addressed, the SDC command provides a method to clear only selected instruments (instead of clearing all instruments simultaneously, as is the case with DCL).

GET (group execute trigger)

GET is a GPIB trigger that is used as an arm event to control operation. The Model 6482 reacts to this trigger if it is the programmed arm control source. The following command selects the GPIB arm control source:

:ARM:SOURce BUS

NOTE   *With :ARM:SOURce BUS selected and an :INITiate command sent, do not send any commands (except GET, DCL, SDC, IFC, *TRG, and :ABORt) while performing source and measure operations (ARM indicator on). If you do, erratic operation will occur.*

SPE, SPD (serial polling)

Use the serial polling sequence to obtain the Model 6482 serial poll byte. The serial poll byte contains important information about internal functions. See Section 13. Generally, the serial polling sequence is used by the controller to determine which of several instruments has requested service with the SRQ line. However, the serial polling sequence may be performed at any time to obtain the status byte from the Model 6482.

Front-panel GPIB operation

This section describes features of the front panel that are part of GPIB operation, including messages, status indicators, and the LOCAL key.

Error and status messages

See Appendix A for a list of status and error messages associated with IEEE-488 programming. The instrument can be programmed to generate a service request (SRQ), and command queries can be performed to check for specific error conditions.

GPIB status indicators

The REM (remote), TALK (talk), LSTN (listen), and SRQ (service request) indicators show the GPIB bus status. Each of these indicators is described below.

REM

This indicator shows when the instrument is in the remote state. REM does not necessarily indicate the state of the bus REN line, because the instrument must be addressed to listen with REN true before the REM indicator turns on. When the instrument is in remote operation, all front-panel keys, except for the LOCAL key, are locked out. When REM is turned off, the instrument is in the local state, and front-panel operation is restored.

NOTE   *If local lockout (LLO) is in effect, LOCAL will be locked out. OUTPUT ON/OFF is still operational in remote mode. If ARM:SOUR is set to manual, the TRIG key will be active during remote operation.*

TALK

This indicator is on when the instrument is in the talker active state. Place the instrument in the talker state by addressing it to talk with the correct MTA (my talk address) command. TALK is off when the instrument is in the talker idle state. Place the instrument in the talker idle state by sending a UNT (untalk) command, addressing it to listen, or sending the IFC (interface clear) command.

LSTN

This indicator is on when the Model 6482 is in the listener active state, which is activated by addressing the instrument to listen with the correct MLA (my listen address) command. LSTN is off when the instrument is in the listener idle state. Place the instrument in the listener idle state by sending UNL (unlisten), addressing it to talk, or sending IFC (interface clear) command over the bus.

SRQ

You can program the instrument to generate a service request (SRQ) when one or more errors or conditions occur. When this indicator is on, a service request has been generated. This indicator

stays on until the serial poll byte is read or all the conditions that caused SRQ have been cleared. See Section 13 for more information.

LOCAL key

The LOCAL key cancels the remote state and restores local operation of the instrument.

Pressing the LOCAL key also turns off the REM indicator and returns the display to normal if a user-defined message was displayed.

If the LLO (local lockout) command is in effect, the LOCAL key is also inoperative.

For safety reasons, the OUTPUT key can be used to turn the output off while in LLO.

Programming syntax

The information in this section covers syntax for both common commands and SCPI commands. For information not covered here, see the IEEE-488.2 and SCPI standards. See Section 14 and Section 16 for more details on common and SCPI commands, respectively.

Command words

Program messages are made up of one or more command words.

Commands and command parameters

Common commands and SCPI commands may or may not use a parameter. The following are some examples:

```
*SAV <NRf>              Parameter (NRf) required
*RST                    No parameter used
:CALCulate1:STATe <b>   Parameter <b> required
:SYSTem:PRESet          No parameter used
```

NOTE   At least one space between the command word and the parameter is required.

Brackets [ ]: Some command words are enclosed in brackets ([ ]). These brackets are used to denote an optional command word that does not need to be included in the program message. For example:

```
:INITiate[:IMMediate]
```

These brackets indicate that :IMMediate is implied (optional) and does not have to be used. Thus, the above command can be sent in one of two ways:

```
:INITiate
```

or

```
:INITiate:IMMediate
```

Notice that the optional command is used without the brackets. When using optional command words in your program, do not include the brackets.

Angle brackets < >: Angle brackets (< >) are used to denote a parameter type. Do not include the brackets in the program message. For example:

`:OUTPut2 <b>`	The <b> indicates a Boolean-type parameter is required. Therefore, to enable the selected source, you must send the command with the ON or 1 parameter as follows:
`:OUTPut2 ON`	
`:OUTPut2 1`	

Parameter types: The following are some of the more common parameter types:

<b>   **Boolean:** Used to enable or disable an instrument operation. 0 or OFF disables the operation, and 1 or ON enables the operation. Example:

`:CALCulate1:STATe ON`   Enable Calc 1 math

<name>   **Name parameter:** Select a parameter name from a listed group. Example:

`<name> = NEVer`

`= NEXT`

`:TRACe:FEED:CONTrol NEXT`

<NRf>   **Numeric representation format:** This parameter is a number that can be expressed as an integer (for example, 8), a real number (for example, 23.6), or an exponent (2.3E6). Example:

`:SYSTem:KEY 11`   Remotely presses the EXIT key

<n>   **Numeric value:** A numeric value parameter can consist of an NRf number or one of the following name parameters: DEFault, MINimum, MAXimum. When the DEFault parameter is used, the instrument is programmed to the *RST default value. When the MINimum parameter is used, the instrument is programmed to the lowest allowable value. When the MAXimum parameter is used, the instrument is programmed to the largest allowable value. Examples:

`:ARM:TIMer 0.1`	Sets timer to 100 ms
`:ARM:TIMer DEFault`	Sets timer to 0.1 s
`:ARM:TIMer MINimum`	Sets timer to 1 ms
`:ARM:TIMer MAXimum`	Sets timer to 99999.99 s

<numlist>   **Numlist:** Specify one or more numbers for a list. Example:

`:STATus:QUEue:ENABle (-110:-222)`

Enable errors -110 through -222

<NDN> **Nondecimal numeric:** This parameter is used to send values in the binary, octal, or hexadecimal format. The prefix designates the format type:

#Bxx...x	#B specifies the binary format. xx...x is the binary number (using zeros (0) and ones (1)).
#Qxx...x	#Q specifies the octal format. xx...x is the octal number (values 0 through 7).
#Hxx...x	#H specifies the hexadecimal format. xx...x is the hexadecimal number (values 0 through 9 and A through F).

Examples to send the decimal value 36 in the nondecimal formats:

`*ESE #b100100`	Binary format
`*ESE #q44`	Octal format
`*ESE #h24`	Hexadecimal format

Query commands

This type of command requests (queries) the presently programmed status. It is identified by the question mark (?) at the end of the fundamental form of the command. Most commands have a query form:

`:ARM:TIMer?`   Queries the timer interval.

Most commands that require a numeric parameter (<n>) can also use the DEFault, MINimum, and MAXimum parameters for the query form. These query forms are used to determine the *RST default value and the upper and lower limits for the fundamental command. Examples are:

`:ARM:TIMer? DEFault`	Queries the *RST default value.
`:ARM:TIMer? MINimum`	Queries the lowest allowable value.
`:ARM:TIMer? MAXimum`	Queries the largest allowable value.

Case sensitivity

Common commands and SCPI commands are not case sensitive. You can use upper or lower case and any case combination. Examples:

```
*RST = *rst
:DATA? = :data?
:SYSTem:PRESet = :system:preset
```

NOTE *Using all upper case will result in slightly faster command response times.*

Long-form and short-form versions

A SCPI command word can be sent in its long-form or short-form version. The command subsystem tables in Section 16 provide the long-form version. However, the short-form version is indicated by upper case characters. Examples:

`:SYSTem:PRESet`	long-form
`:SYST:PRES`	short-form
`:SYSTem:PRES`	long-form and short-form combination

Note that each command word must be in long-form or short-form, and not something in between. For example, :SYSTe:PRESe is illegal and will generate an error. The command will not be executed.

Short-form rules

Use the following rules to determine the short-form version of any SCPI command:

- If the length of the command word is four letters or less, no short form version exists. Example:

  `:AUTO = :AUTO`

These rules apply to command words that exceed four letters:

- If the fourth letter of the command word is a vowel (including "y"), delete it and all the letters after it. Example:

  `:IMMediate = :IMM`

- If the fourth letter of the command word is a consonant, retain it but drop all the letters after it. Example:

  `:FORMat = :FORM`

- If the command contains a question mark (?; query) or a non-optional number included in the command word, you must include it in the short-form version. Example:

  `:DELay? = :DEL?`

- Command words or characters that are enclosed in brackets ([ ]) are optional and need not be included in the program message.

NOTE   *For fastest response to commands, always use short forms.*

Program messages

A program message is made up of one or more command words sent by the computer to the instrument. Each common command is a three letter acronym preceded by an asterisk (*). The following example uses the SCPI commands from the :STATus subsystem to explain how command words are structured to formulate program messages.

`:STATus`	Path (root)
`    :OPERation`	Path
`        :ENABle <NRf>`	Command and parameter
`        :ENABle?`	Query command
`    :PRESet`	Command

Single command messages

The above command structure has three levels. The first level is made up of the root command (:STATus) and serves as a path. The second level is made up of another path (:OPERation) and a command (:PRESet). The third path is made up of one command for the :OPERation path. The three commands in this structure can be executed by sending three separate program messages as follows:

```
:STAT:OPER:ENAB <NRf>
:STAT:OPER:ENAB?
:STAT:PRES
```

In each of the above program messages, the path pointer starts at the root command (:STAT) and moves down the command levels until the command is executed.

Multiple command messages

You can send multiple command messages in the same program message if they are separated by semicolons (;). The following is an example showing two commands in one program message:

```
:STAT:OPER; :STAT:OPER:ENAB <NRf>
```

When the above message is sent, the first command word is recognized as the root command (:STAT). When the next colon is detected, the path pointer moves down to the next command level and executes the command. When the path pointer sees the colon after the semicolon (;), it resets back to the root level and starts over.

Commands that are on the same command level can be executed without having to retype the entire command path. Example:

```
:STAT:OPER:ENAB <NRf>; ENAB?
```

After the first command (:ENAB) is executed, the path pointer is at the third command level in the structure. Since :ENAB? is also on the third level, it can be typed in without repeating the entire path name. Notice that the leading colon for :ENAB? is not included in the program message. If a colon were included, the path pointer would reset to the root level and expect a root command. Because :ENAB? is not a root command, an error would occur.

Command path rules

- Each new program message must begin with the root command, unless it is optional (for example, [:SENSe[1]]). If the root is optional, the command word on the next level is the root. For fastest operation, do not send optional data.
- The colon (:) at the beginning of a program message is optional and need not be used. Note that eliminating the first colon will result in fastest operation. Example:

  ```
  :STAT:PRES = STAT:PRES
  ```

- When a colon (:) is detected in the command path, the path is changed to the next command level. An exception is when the path pointer detects a semicolon (;), which is used to separate commands within the program message (see the following rule).
- When a colon (:) that immediately follows a semicolon (;) is detected, it resets to the root level.
- Detection can only move down a path; it cannot be moved up a level. Executing a command at a higher level requires that you start over at the root command.

Using common and SCPI commands in the same message

Both common commands and SCPI commands can be used in the same message if they are separated by semicolons (;). A common command can be executed at any command level and will not affect the path pointer. Example:

```
:STAT:OPER:ENAB <NRf>; *ESE <NRf>
```

Program message terminator (PMT)

Each program message must be terminated with an LF (line feed), EOI (end or identify), or an LF+EOI. The bus will stop responding if your computer does not provide this termination. The following example shows how a multiple-command program message must be terminated:

```
:OUTP1 ON <PMT>
```

Command execution rules

- Commands execute in the order that they are presented in the program message.
- An invalid command generates an error and is not executed.
- Valid commands that precede an invalid command in a multiple-command program message are executed.
- Valid commands that follow an invalid command in a multiple-command program message are ignored.

Response messages

A response message is the message sent by the instrument to the computer in response to a query command program message.

Sending a response message

After sending a query command, the response message is placed in the output queue. When the Model 6482 is addressed to talk, the response message is sent from the output queue to the computer.

Multiple response messages

If you send more than one query command in the same program message (Multiple command messages), the multiple response messages for all the queries are sent to the computer when the Model 6482 is addressed to talk. The responses are sent in the order the query commands were sent and are separated by semicolons (;). Items within the same query are separated by commas (,). The following example shows the response message for a program message that contains four single-item query commands:

```
0; 1; 1; 0
```

Response message terminator (RMT)

Each response is terminated with an LF (line feed) and EOI (end or identify). The following example shows how a multiple response message is terminated:

```
0; 1; 1; 0 <RMT>
```

Message exchange protocol

Two rules summarize the message exchange protocol:

Rule 1. You must always tell the Model 6482 what to send to the computer.

The following two steps must always be performed to send information from the instrument to the computer:

1. Send the appropriate query commands in a program message.
2. Address the Model 6482 to talk.

Rule 2. The complete response message must be received by the computer before another program message can be sent to the Model 6482.

RS-232 interface operation

> **NOTE**   *RS-232 operation (baud rate, data bits, parity, and terminator) is configured from the COMMUNICATION option of the main menu. See Section 1, "Main menu."*

Sending and receiving data

The RS-232 interface transfers data using 8 data bits, 1 stop bit, and no parity. Make sure the device you connect to the Model 6482 also uses these settings.

You can break data transmissions by sending a ^C (decimal 3) or ^X (decimal 18) character string to the instrument, or by sending an RS-232 break condition (holding the transmit line low for >11 bits). This clears any pending operation, discards any pending output, and returns a device clear (DCL) message.

Baud rate

The baud rate is the rate at which the Model 6482 and the programming terminal communicate. Choose one these available rates:

- 57600
- 38400
- 19200
- 9600
- 4800
- 2400
- 1200
- 600
- 300

The factory default baud rate is 9600.

When you choose a baud rate, make sure the programming terminal or printer that you are connecting to the Model 6482 can support the baud rate you selected. Both the Model 6482 and the other device must be configured for the same baud rate.

Data bits and parity

The RS-232 interface can be configured to send and receive data that is 7 or 8 bits long using even, odd, or no parity. No parity is only valid when using 8 data bits.

Terminator

The Model 6482 can be configured to terminate each program message that it transmits to the controller with any of the following combinations of <CR> and <LF>:

<CR>   Carriage return
<CR+LF>   Carriage return and line feed
<LF>   Line feed
<LF+CR>   Line feed and carriage return

Flow control (signal handshaking)

Signal handshaking between the controller and the instrument lets the two devices communicate with each other about readiness to receive data. The Model 6482 supports software flow control (signal handshaking); it does not support hardware handshaking (flow control).

Software flow control is in the form of XON and XOFF characters. This flow control is enabled when XON-XOFF is selected from the RS-232 FLOW CONTROL menu. When the input queue of the instrument becomes more than 3/4 full, the instrument issues an XOFF character. The control program should respond to this and stop sending characters until the Model 6482 issues the XON character (which it will do when its input buffer has decreased below half-full). The Model 6482 recognizes XON and XOFF characters sent from the controller. An XOFF character will cause the instrument to stop outputting characters until it receives an XON character. Incoming commands are processed after the <CR> character is received from the controller.

If NONE is the selected flow control, there will be no signal handshaking between the controller and the Model 6482. Data will be lost if transmitted before the receiving device is ready.

RS-232 connections

The RS-232 serial port (Figure 12-3) is connected to the serial port of a computer using a straight-through RS-232 cable terminated with DB-9 connectors. Do not use a null modem cable. The

serial port uses the transmit (TXD), receive (RXD), and signal ground (GND) lines of the RS-232 standard. Figure 12-4 shows the rear panel connector for the RS-232 interface, and Table 12-2 shows the pinout for the connector.

If your computer uses a DB-25 connector for the RS-232 interface, you will need a cable or adapter with a DB-25 connector on one end, and a DB-9 connector on the other. Make sure the cable is wired straight through (not null modem).

Figure 12-4
RS-232 interface connector

RS-232

5 4 3 2 1

9 8 7 6
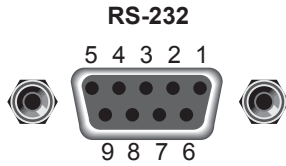
Rear panel connector

Table 12-2
RS-232 connector pinout

Pin number	Description
1	Not used
2	TXD, transmit data
3	RXD, receive data
4	Not used
5	GND, signal ground
6	Not used
7	RTS, ready to send
8	CTS, clear to send
9	Not used

Note: CTS and RTS are tied together.
      Pins 1, 4, and 6 are tied together.

Table 12-3 provides pinout identification for the 9-pin (DB-9) or 25-pin (DB-25) serial port connector on the computer.

Table 12-3
Computer serial port pinout

Signal	DB-9 pin number	DB-25 pin number
DCD, data carrier detect	1	8
RXD, receive data	2	3
TXD, transmit data	3	2
DTR, data terminal ready	4	20
GND, signal ground	5	7
DSR, data set ready	6	6
RTS, request to send	7	4
CTS, clear to send	8	5
RI, ring indicator	9	22

Error messages

See Appendix A for RS-232 error messages.

This page left blank intentionally.

In this section:

Overview

Each Keithley Instruments Model 6482 provides a number of status registers and queues that are collectively referred to as the "status model." Through manipulation and monitoring of these registers and queues, you can view and control various instrument events. You can include commands in your test program that can determine if a service request (SRQ) event has occurred and the cause of the event.

The status structure is shown in Figure 13-1. The center of the status structure is the status byte register; all status model registers and queues flow into the Status Byte Register.

Status byte and SRQ

The status byte register receives the summary bits of four status register sets and two queues. The register sets and queues monitor the various instrument events. When an enabled event occurs, it sets a summary bit in the status byte register. When a summary bit of the status byte is set and its corresponding enable bit is set (as programmed by the user), the request for service/ master summary status (RQS/MSS) bit will set to indicate that an SRQ has occurred.

Status register sets

A typical status register set is made up of a condition register, an event register, and an event enable register. A condition register is a read-only register that constantly updates to reflect the present operating conditions of the instrument.

When an event occurs, the appropriate event register bit sets to 1. The bit remains latched to 1 until the register is reset. When an event register bit is set and its corresponding enable bit is set (as programmed by the user), the output (summary) bit of the register will set to 1, which then sets the summary bit of the status byte register.

Queues

The Model 6482 uses an output queue and an error queue. The response messages to query commands are placed in the output queue. As various programming errors and status messages occur, they are placed in the error queue. When a queue contains data, it sets the appropriate summary bit of the status byte register.

Figure 13-1
Model 6482 status register structure
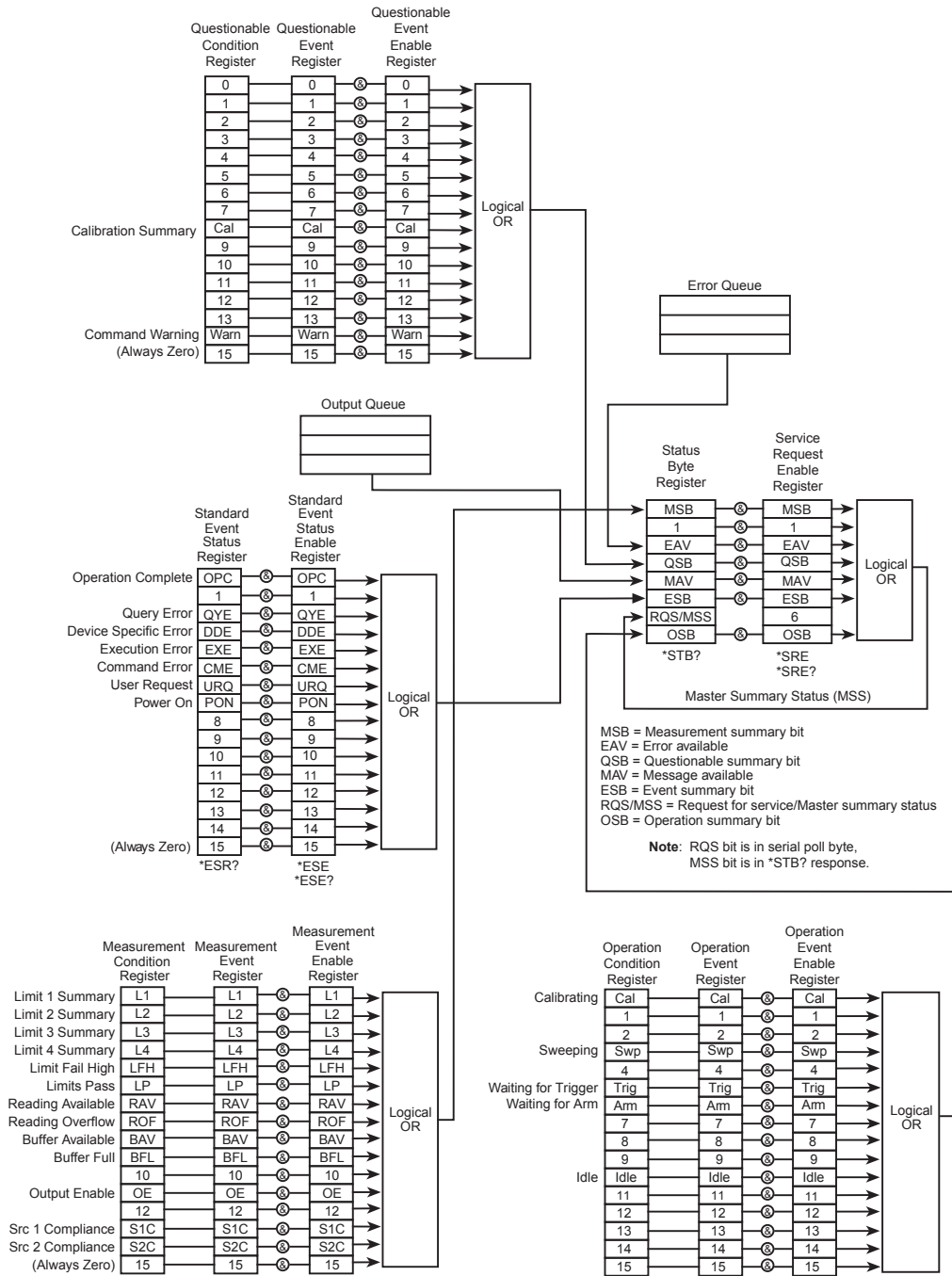
Clearing registers and queues

When the Model 6482 is turned on, the bits of all registers in the status structure are cleared (reset to 0), and the two queues are empty. Commands to reset the event and event enable registers, and the error queue are listed in Table 13-1. In addition to these commands, any enable register can be reset by sending a parameter value of zero (0) with the individual command to program the register.

NOTE   SYSTem:PRESet and *RST have no effect on status structure registers and queues.

Table 13-1
Common and SCPI commands to reset registers and clear queues

Commands	Description	Note reference

To reset the registers:
`*CLS`
	Reset all bits of the following event registers to 0:
	    Standard Event Register
	    Operation Event Register
	    Measurement Event Register
	    Questionable Event Register
		Note 1

`:STATus:PRESet`	Reset all bits of the following enable registers to 0:
	    Operation Event Enable Register
	    Measurement Event Enable Register
	    Questionable Event Enable Register
		Note 1

To clear the error queue: `*CLS`	Clear all messages from Error Queue	Note 2
`:STATus:QUEue:CLEar`	Clear messages from Error Queue	Note 3
`:SYSTem:ERRor:CLEar`	Clear messages from Error Queue	Note 3

Notes:
1. The Standard Event Enable Register is not reset by STATus:PRESet or *CLS. Send the *ESE command with a zero (0) parameter value to reset all bits of that enable register to 0. See "Status byte and service request commands," page Section 13-8.
2. STATus:PRESet has no effect on the error queue.
3. Use either of the two clear commands to clear the error queue.

Programming and reading registers

This topic explains how to program enable registers and read any register in the status structure.

Programming enable registers

The only registers that can be programmed by the user are the enable registers. All other registers in the status structure are read-only registers. The following topics explain how to determine the parameter values for the various commands used to program enable registers. Details about the actual commands are covered later in this section (Table 13-3 and Table 13-6).

A command to program an event enable register is sent with a parameter value that sets the state (0 or 1) of each bit in the appropriate register. An enable register can be programmed using any of the following data formats for the parameter value: Binary, decimal, hexadecimal, or octal.

The bit positions of the register (Figure 13-2) indicate the binary parameter value. For example, if you wish to set bits B4, B3, and B1, the binary value would be 11010 (where B4=1, B3=1, B1=1, and all other bits are 0). When you use one of the other formats, convert the binary number to its decimal, hexadecimal, or octal equivalent:

Binary 11010 = Decimal 26 = Hexadecimal 1A = Octal 32

Note that Figure 13-2 includes the decimal weight for each register bit. To set bits B4, B3, and B1, the decimal parameter value would be the sum of the decimal weights for those bits (16+8+2 = 26).

Figure 13-2
16-bit status register

Bit position	B7	B6	B5	B4	B3	B2	B1	B0
Binary value	0/1	0/1	0/1	0/1	0/1	0/1	0/1	0/1
Decimal	128	64	32	16	8	4	2	1
Weights	(2^7)	(2^6)	(2^5)	(2^4)	(2^3)	(2^2)	(2^1)	(2^0)

A. Bits 0 through 7

Bit position	B15	B14	B13	B12	B11	B10	B9	B8
Binary value	0/1	0/1	0/1	0/1	0/1	0/1	0/1	0/1
Decimal	32768	16384	8192	4096	2048	1024	512	256
Weights	(2^{15})	(2^{14})	(2^{13})	(2^{12})	(2^{11})	(2^{10})	(2^9)	(2^8)

B. Bits 8 through 15

The <NDN> (nondecimal numeric) parameter type is used to send nondecimal values. These values require a header (#B, #H, or #Q) to identify the data format being sent. The letter in the header can be upper or lower case. The <NRf> (numeric representation format) parameter type is used to send decimal values, and does not use a header. The following examples show the proper parameter syntax for setting bits B5, B3, and B2:

#b101100	Binary format (<NDN> parameter type)
#h2C	Hexadecimal format (<NDN> parameter type)
#q54	Octal format (<NDN> parameter type)
44	Decimal format (<NRf> parameter type)

Valid characters for the non-decimal parameter values are shown as follows:

<NDN> format	Valid characters
Binary	1 and 0
Hexadecimal	0 through 9 and A through F
Octal	0 through 7

Reading registers

Any register in the status structure can be read by using the appropriate query (?) command. The following explains how to interpret the returned value (response message). The actual query commands are covered later in this section (Table 13-3 through Table 13-7).

The response message will be a value that indicates which bits in the register are set. That value (if not already binary) must be converted to its binary equivalent. For example, for a binary value of 100101, bits B5, B2, and B0 are set.

The returned value can be in the binary, decimal, hexadecimal, or octal format. The FORMat:SREGister command is used to select the data format for the returned value (Table 13-2).

For nondecimal formats, one of the following headers will accompany the returned value to indicate which format is selected:

#B = Header for binary values
#H = Header for hexadecimal values
#Q = Header for octal values

Table 13-2
Data format commands for reading status registers

Command	Description	Default

`:FORMat:SREGister <name>`	Select data format for reading status registers:
	<name> = ASCiiDecimal format
	           HEXadecimalHexadecimal format
	           OCTalOctal format
	           BINaryBinary format
		ASCii

Status byte and service request (SRQ)

Service request is controlled by two 8-bit registers; the status byte register and the service request enable register. Figure 13-3 shows the structure of these registers.

Figure 13-3
Status byte and service request (SRQ)
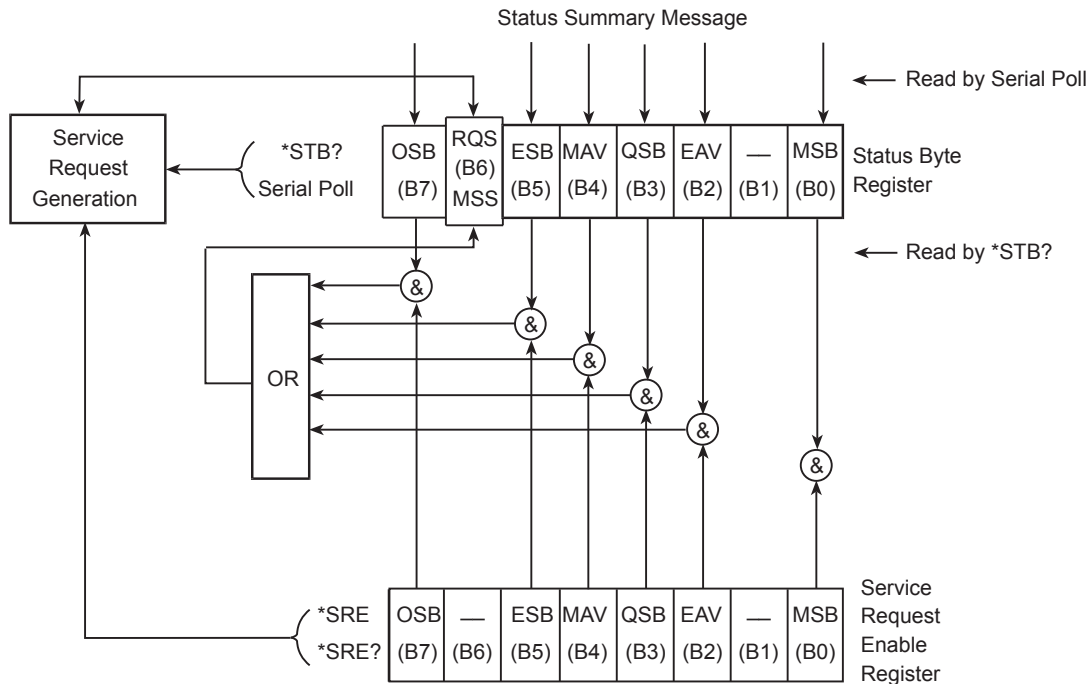
OSB = Operation summary bit
MSS = Master summary status
RQS = Request for service
ESB = Event summary bit
MAV = Message available
QSB = Questionable summary bit
EAV = Error available
MSB = Measurement summary bit
&   = Logical AND
OR = Logical OR

Status byte register

The summary messages from the status registers and queues are used to set or clear the appropriate bits (B0, B2, B3, B4, B5, and B7) of the status byte register. These summary bits do not latch, and their states (0 or 1) are solely dependent on the summary messages (0 or 1). For example, if the standard event register is read, its register will clear. As a result, its summary message will reset to 0, which then resets the ESB bit in the status byte register.

The bits of the status byte register are described as follows:

- **Bit B0, Measurement Summary Bit (MSB)**: Set summary bit indicates that an enabled measurement event has occurred.
- **Bit B1**: Not used.
- **Bit B2, Error Available (EAV)**: Set summary bit indicates that an error or status message is present in the Error Queue.
- **Bit B3, Questionable Summary Bit (QSB)**: Set summary bit indicates that an enabled questionable event has occurred.
- **Bit B4, Message Available (MAV)**: Set summary bit indicates that a response message is present in the Output Queue.
- **Bit B5, Event Summary Bit (ESB)**: Set summary bit indicates that an enabled standard event has occurred.
- **Bit B6, Request Service (RQS)/Master Summary Status (MSS)**: Set bit indicates that an enabled summary bit of the Status Byte Register is set.
- **Bit B7, Operation Summary (OSB)**: Set summary bit indicates that an enabled operation event has occurred.

Depending on how it is used, bit B6 of the Status Byte Register is either the Request for Service (RQS) bit or the Master Summary Status (MSS) bit:

- When using the serial poll sequence of the Model 6482 to obtain the status byte (serial poll byte), bit B6 is the RQS bit. See "Serial polling and SRQ" for details on using the serial poll sequence.
- When using the *STB? command to read the status byte (see Table 13-3), bit B6 is the MSS bit.

Service request enable register

The generation of a service request is controlled by the service request enable register. This register is programmed by you and is used to enable or disable the setting of bit B6 (RQS/MSS) by the status summary message bits (B0, B2, B3, B4, B5, and B7) of the status byte register. As shown in Figure 13-3, a logical AND (&) operation is performed on the summary bits and the corresponding enable bits of the service request enable register. When a set (1) summary bit has a logical AND (&) operation performed with an enabled (1) bit of the enable register, the logic "1" output is applied to the input of the OR gate and, therefore, sets the MSS/RQS bit in the status byte register.

The individual bits of the service request enable register can be set or cleared by using the *SRE common command. To read the service request enable register, use the *SRE? query command. The service request enable register clears when the instrument is restarted or a when parameter value of 0 is sent with the *SRE command (for example, *SRE 0). The commands to program and read the service request enable register are listed in Table 13-3.

Serial polling and SRQ

Any enabled event summary bit that goes from 0 to 1 will set bit B6 and generate an SRQ (service request). In your test program, you can periodically read the status byte to check if an SRQ has occurred and what caused it. If an SRQ occurs, the program can, for example, branch to an appropriate subroutine that will service the request.

Typically, SRQs are managed by the serial poll sequence of the Model 6482. If an SRQ does not occur, bit B6 (RQS) of the status byte register will remain cleared, and the program will simply proceed normally after the serial poll is performed. If an SRQ does occur, bit B6 of the status byte register will set, and the program can branch to a service subroutine when the SRQ is detected by the serial poll.

The serial poll automatically resets RQS of the status byte register. This allows subsequent serial polls to monitor bit B6 for an SRQ occurrence generated by other event types. After a serial poll, the same event can cause another SRQ, even if the event register that caused the first SRQ has not been cleared.

The serial poll does not clear the master summary status bit (MSS). The MSS bit stays set until all status byte summary bits are reset.

SPE, SPD (serial polling)

The SPE, SPD general bus command sequence is used to serial poll the Model 6482. Serial polling obtains the serial poll byte (status byte). Typically, serial polling is used by the controller to determine which of several instruments has requested service with the SRQ line.

Status byte and service request commands

The commands to program and read the status byte register and service request enable register are listed in Table 13-3. For details on programming and reading registers, see "Programming enable registers," page Section 13-4, and "Reading registers," page Section 13-5.

NOTE   *To reset the bits of the service request enable register to 0, use 0 as the parameter value for the *SRE command (for example, *SRE 0).*

Table 13-3
Status byte and service request enable register commands

Command	Description

`*STB?`
`*SRE <NDN> or <NRf>`
 	
`*SRE?`
	Read Status Byte Register.
	Program the Service Request Enable Register:
	   <NDN> = #Bxx…x Binary format (each x = 1 or 0)
	            = #Hx Hexadecimal format (x = 0 to FF)
	            = #Qx Octal format (x = 0 to 377)
	   <NRf> = 0 to 255 Decimal format
	Read the Service Request Enable Register

Note: *CLS and STATus:PRESet have no effect on the service request enable register.

Programming example: Set MSS (B6) when error occurs

The first commands of the sequence in Table 13-4 clears the error queue and programs the service request enable register to 4. When an invalid command is sent (line 4), bits B2 (EAV) and B6 (MSS) of the status byte register set to 1. The last command reads the status byte register using the binary format (which directly indicates which bits are set). The command to select format (FORMat:SREGister) is documented in Table 13-2. To determine the exact nature of the error, you will have to read the error queue. Refer to "Queues."

Table 13-4
Status byte programming example

Command	Description

`*CLS`
`*SRE 4`
`FORM:SREG BIN`
`*XYZ`
`*STB?`
	Clear error queue.
	Programs service request enable register
	Select binary format.
	Generate error.
	Read status byte register.

Status register sets

As shown in Figure 13-1, there are four status register sets in the status structure of the Model 6482: Standard Event Status, Operation Event Status, Measurement Event Status, and Questionable Event Status.

NOTE   *See Appendix A for details on which register bits are set by specific error and status conditions.*

Register bit descriptions

Standard event register

The used bits of the standard event register (shown in Figure 13-4) are described as follows:

Figure 13-4
Standard event status

PON = Power on — DDE = Device-dependent error
URQ = User request — QYE = Query error
CME = Command error — OPC = Operation complete
EXE = Execution error — & = Logical AND
— OR = Logical OR
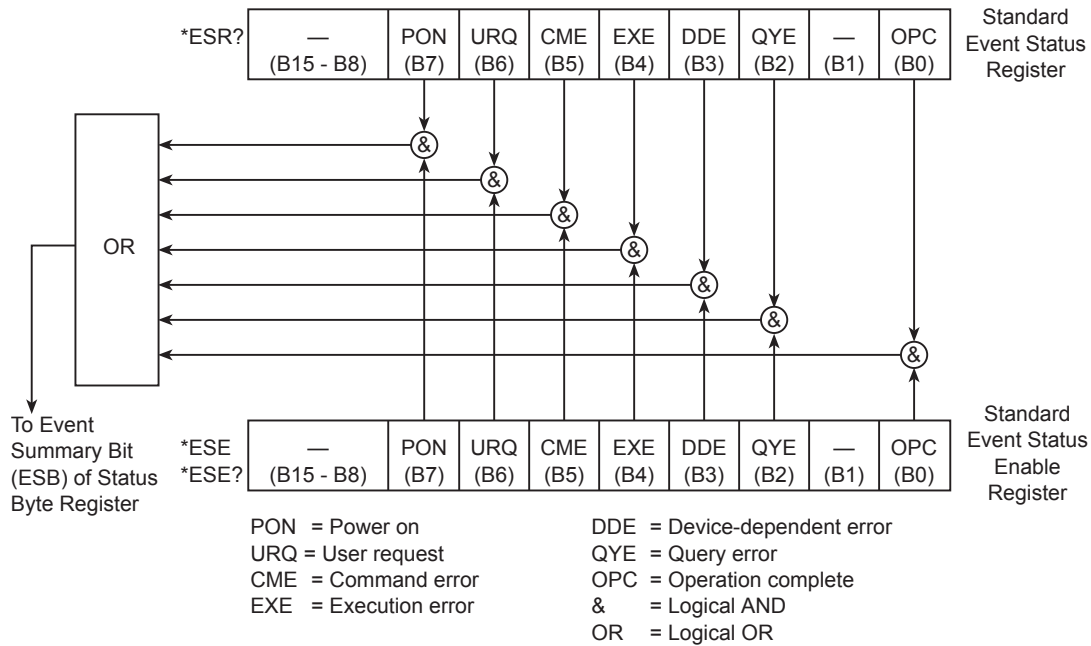
- **Bit B0, Operation Complete (OPC)**: Set bit indicates that all pending selected device operations are completed and the Model 6482 is ready to accept new commands. This bit only sets in response to the *OPC? query command. See Section 14 for details on *OPC and *OPC?.
- **Bit B1**: Not used.
- **Bit B2, Query Error (QYE)**: Set bit indicates that you attempted to read data from an empty output queue.
- **Bit B3, Device-Dependent Error (DDE)**: Set bit indicates that an instrument operation did not execute properly because of an internal condition.
- **Bit B4, Execution Error (EXE)**: Set bit indicates that the Model 6482 detected an error while trying to execute a command.
- **Bit B5, Command Error (CME)**: Set bit indicates that a command error has occurred.
- Command errors include:
  - IEEE-488.2 syntax error: Model 6482 received a message that does not follow the defined syntax of the IEEE-488.2 standard.
  - Semantic error: Model 6482 received a command that was misspelled or received an optional IEEE-488.2 command that is not implemented.
  - The instrument received a group execute trigger (GET) inside a program message.
- **Bit B6, User Request (URQ)**: Set bit indicates that the LOCAL key on the Model 6482 front panel was pressed.
- **Bit B7, Power ON (PON)**: Set bit indicates that the Model 6482 has been turned off and turned back on since the last time this register was read.

Operation event register

The used bits of the operation event register (Figure 13-5) are described as follows:

- **Bit B0, Calibrating (Cal)**: Set bit indicates that the Model 6482 is calibrating.
- **Bits B1** through **B4**: Not used.
- **Bit B5, Waiting for Trigger Event (Trig)**: Set bit indicates that the Model 6482 is in the trigger layer waiting for a TLINK trigger event to occur.
- **Bit B6, Waiting for Arm Event (Arm)**: Set bit indicates that the Model 6482 is in the arm layer waiting for an arm event to occur.
- **Bits B7** through **B9**: Not used.
- **Bit B10, Idle State (Idle)**: Set bit indicates the Model 6482 is in the idle state.
- **Bits B11** through **B15**: Not used.

Figure 13-5
Operation event status
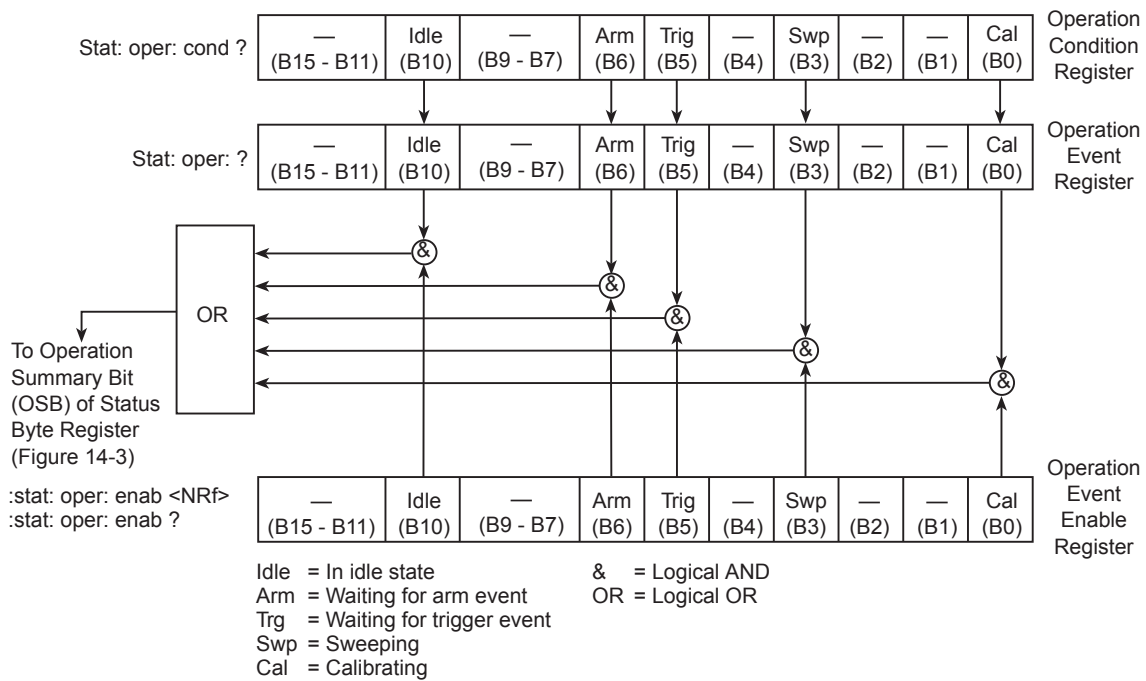
Measurement event register

The used bits of the measurement event register (Figure 13-6) are described as follows:

- **Bit B0, Limit 1 Summary (L1)**: Set bit indicates Limit 1 summary information is available in the :FORM:ELEM STAT element (see Section 16, "FORMat subsystem").
- **Bit B1, Limit 2 Summary (L2)**: Set bit indicates Limit 2 summary information is available in the :FORM:ELEM STAT element.
- **Bit B2, Limit 3 Summary (L3)**: Set bit indicates Limit 3 summary information is available in the :FORM:ELEM STAT element.
- **Bit B3, Limit 4 Summary (L4)**: Set bit indicates Limit 4 summary information is available in the :FORM:ELEM STAT element.
- **Bit B4, Limit Fail High (LFH)**: Set bit indicates that one of the high limit tests has failed.
- **Bit B5, Limits Pass (LP)**: Set bit indicates that all limit tests passed.
- **Bit B6, Reading Available (RAV)**: Set bit indicates that a reading was taken and processed.
- **Bit B7, Reading Overflow (ROF)**: Set bit indicates that the current reading exceeds the selected measurement range of the Model 6482.
- **Bit B8, Buffer Available (BAV)**: Set bit indicates that there are at least two readings in the buffer.
- **Bit B9, Buffer Full (BFL)**: Set bit indicates that the trace buffer is full.
- **Bit B10**: Not used.
- **Bit B11, Output Enable Asserted (OE)**: Set bit indicates that the output enable line is at digital low (asserted). Source outputs can be turned on.
- **Bit B12**: Not used.
- **Bit B13, Source 1 Compliance (S1C)**: Set bit indicates that the channel 1 voltage source is in compliance.
- **Bit B14, Source 2 Compliance (S2C)**: Set bit indicates that the channel 2 voltage source is in compliance.
- **Bit B15**: Not used.

Figure 13-6
Measurement event status

	S2C	S1C		OE		BFL	BAV	ROF	RAV	LP	LFH	L4	L3	L2	L1	

STAT:MEAS:COND? — (B15), S2C (B14), S1C (B13), — (B12), OE (B11), — (B10), BFL (B9), BAV (B8), ROF (B7), RAV (B6), LP (B5), LFH (B4), L4 (B3), L3 (B2), L2 (B1), L1 (B0) — Measurement Condition Register

STAT:MEAS? — (B15), S2C (B14), S1C (B13), — (B12), OE (B11), — (B10), BFL (B9), BAV (B8), ROF (B7), RAV (B6), LP (B5), LFH (B4), L4 (B3), L3 (B2), L2 (B1), L1 (B0) — Measurement Event Register

OR

To Measurement Summary Bit (OSB) of Status Byte Register

STAT:MEAS:ENAB <NRf>
STAT:MEAS:ENAB? — (B15), S2C (B14), S1C (B13), — (B12), OE (B11), — (B10), BFL (B9), BAV (B8), ROF (B7), RAV (B6), LP (B5), LFH (B4), L4 (B3), L3 (B2), L2 (B1), L1 (B0) — Measurement Event Enable Register

S2C = Source 2 in compliance
S1C = Source 1 in compliance
OE = Output enable asserted
BFL = Buffer full
BAV = Buffer available
ROF = Reading overflow
RAV = Reading available

LP = Limits pass
LFH = Limit failed high
L4 = Limit 4 summary
L3 = Limit 3 summary
L2 = Limit 2 summary
L1 = Limit 1 summary

& = Logical AND
OR = Logical OR

Questionable event register

The used bits of the questionable event register (Figure 13-7) are described as follows:

- **Bits B0** through **B7**: Not used.
- **Bit B8, Calibration Summary (Cal)**: Set bit indicates that an invalid calibration constant was detected during the power-up sequence. This error will clear after successful calibration of the instrument.
- **Bits B9** through **B13**: Not used.
- **Bit B14, Command Warning (Warn)**: Set bit indicates that a signal oriented measurement command parameter has been ignored.
- **Bit B15**: Not used.

Figure 13-7
Questionable event status
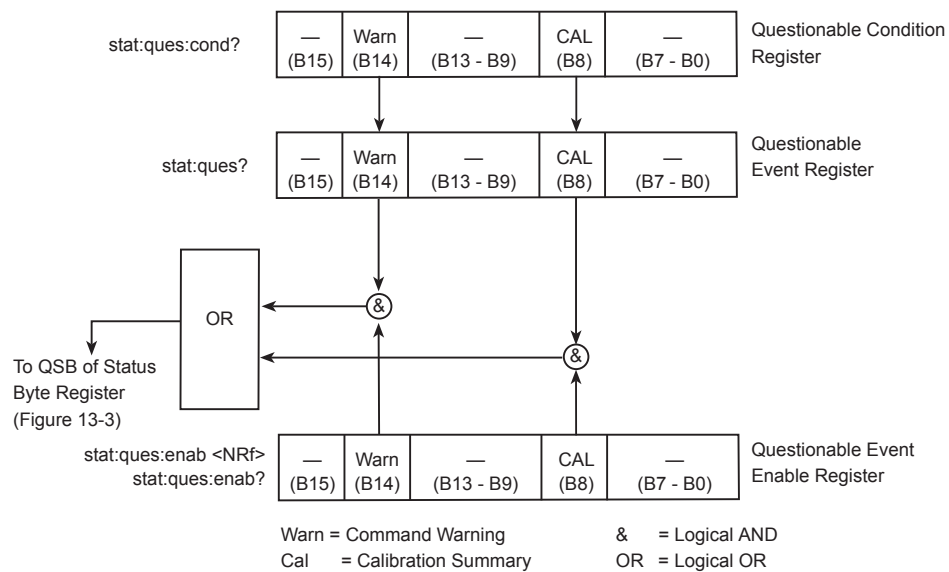
Condition registers

As Figure 13-1 shows, each status register set (except the standard event register set) has a condition register. A condition register is a real-time, read-only register that constantly updates to reflect the present operating conditions of the instrument. For example, while the Model 6482 is in the idle state, bit B10 (idle) of the operation condition register will be set. When the instrument is taken out of the idle state, bit B10 clears.

The commands to read the condition registers are listed in Table 13-5. For details on reading registers, see "Reading registers," page Section 13-5.

Table 13-5
Condition register commands

Command	Description

`:STATus:OPERation:CONDition?`
`:STATus:MEASurement:CONDition?`
`:STATus:QUEStionable:CONDition?`
	Read Operation Condition Register.
	Read Measurement Condition Register.
	Read Questionable Condition Register.

Event registers

As Figure 13-1 shows, each status register set has an event register. When an event occurs, the appropriate event register bit sets to 1. The bit remains latched to 1 until the register is reset. Reading an event register clears the bits of that register. *CLS resets all four event registers.

The commands to read the event registers are listed in Table 13-6. For details on reading registers, see "Reading registers," page Section 13-5.

Table 13-6
Event register commands

Command	Description

`*ESR?`
`:STATus:OPERation:[:EVENt]?`
`:STATus:MEASurement:[:EVENt]?`
`:STATus:QUEStionable:[:EVENt]?`
	Read Standard Event Status Register.
	Read Operation Event Register.
	Read Measurement Event Register.
	Read Questionable Event Register.

Note: Power-up and *CLS resets all bits of all event registers to 0. STATus:PRESet has no effect.

Event enable registers

As Figure 13-1 shows, each status register set has an enable register. Each event register bit as a logical AND (&) operation to a corresponding enable bit of an enable register performed on it. Therefore, when an event bit is set and the corresponding enable bit is set (as programmed by the user), the output (summary) of the register will set to 1, which in turn sets the summary bit of the status byte register.

The commands to program and read the event enable registers are listed in Table 13-7. For details on programming and reading registers, see "Programming enable registers," page Section 13-4 and "Reading registers," page Section 13-5.

> *NOTE* *The bits of any enable register can be reset to 0 by sending the 0 parameter value with the appropriate enable command (for example, STATus:OPERation:ENABle 0).*

Table 13-7
Event enable register commands

Command	Description	Default
`*ESE <NDN> or <NRf>` `*ESE?`	Program Standard Event Enable Register. See "Parameters." Read Standard Event Enable Register.	Note

`STATus`
` :OPERation`
`  :ENABle <NDN> or <NRf>`
`  :ENABle?`
` :MEASurement`
`  :ENABle <NDN> or <NRf>`
`  :ENABle?`
` :QUEStionable`
`  :ENABle <NDN> or <NRf>`
`  :ENABle?`
	STATus Subsystem:
	  Operation Event Enable Register:
	    Program enable register. See "Parameters."
	    Read enable register.
	  Measurement Event Enable Register:
	    Program enable register. See "Parameters."
	    Read enable register.
	  Questionable Event Enable Register:
	    Program enable register. See "Parameters."
	    Read Measurement Event Enable Register.	

Parameters:
 <NDN>=#Bxx…x Binary format (each x = 1 or 0)
  = #Hx Hexadecimal format (x = 0 to FFFF)
  = #Qx Octal format (x = 0 to 177777)
 <NRf>=0 to 65535 Decimal format		

Note: Power-up and STATus:PRESet resets all bits of all enable registers to 0. *CLS has no effect.

Programming example - program and read register set

The command sequence in Table 13-8 programs and reads the measurement register set. Registers are read using the binary format (which directly indicates which bits are set). The command to select format (FORMat:SREGister) is documented in Table 13-2.

Table 13-8
Program and read register programming example

Command	Description

`FORM:SREG BIN`
`STAT:MEAS:ENAB 512`
`STAT:MEAS:COND?`
`STAT:MEAS?`
	Select binary format to read registers.
	Enable BFL (buffer full).
	Read Measurement Condition Register.
	Read Measurement Event Register.

Queues

The Model 6482 uses two queues, which are first-in, first-out (FIFO) registers:

- **Output Queue**: Used to hold reading and response messages.
- **Error Queue**: Used to hold error and status messages. For a listing of error and status messages, see Appendix A.

The Model 6482 status model (Figure 13-1) shows how the two queues are structured with the other registers.

Output queue

The output queue holds data that pertains to the normal operation of the instrument. For example, when a query command is sent, the response message is placed in the output queue.

When data is placed in the output queue, the Message Available (MAV) bit in the status byte register sets. A data message is cleared from the output queue when it is read. The output queue is considered cleared when it is empty. An empty output queue clears the MAV bit in the status byte register.

A message is read from the output queue by addressing the Model 6482 to talk after the appropriate query is sent.

Error queue

The error queue holds error and status messages. As programming errors and status messages occur, a message that defines the error or status is placed in the error queue.

When a message is placed in the error queue, the error available (EAV) bit in the status byte register is set. An error or status message is cleared from the error queue when it is read. The error queue is considered cleared when it is empty. An empty error queue clears the EAV bit in the status byte register.

The error queue holds up to 10 error and status messages. The commands to read the error queue are listed in Table 13-9. When you read a single message in the error queue, the oldest message is read and then removed from the queue. If the queue becomes full, the error code 350, "Queue Overflow," will occupy the last memory location. On power-up, the error queue is empty. When empty, the message "0, No Error" is placed in the queue.

Messages in the error queue are preceded by a code number. Negative (-) numbers are used for SCPI-defined messages, and positive (+) numbers are used for Keithley-defined messages. The messages are listed in Appendix A. As shown in Table 13-9, there are commands to read the entire message (code and message), or the code only.

On power-up, all error messages are enabled and will go into the error queue as they occur. Status messages are not enabled and will not go into the queue. As listed in Table 13-9, there are commands to enable and disable messages. For these commands, the <list> parameter is used to specify which messages to enable or disable. The messages are specified by their codes. The following examples show various forms for using the <list> parameter.

<list> = (-110)   Single message
    = (-110:-222)   Range of messages (-110 through -222)
    = (-110:-222, -220) Range entry and single entry (separated by a comma)

When you enable specific messages, messages not specified in the list are disabled. When you disable messages, each listed message is removed from the enabled list.

NOTE   *To prevent all messages from entering the error queue, send the enable command with the null list parameter as follows: STATus:QUEue:ENABle ().*

Table 13-9
Error queue commands

Command	Description	Comment
STATus	STATus subsystem:	
:QUEue	Read error queue:	Note 1
[:NEXT]?	Read and clear oldest error and status (code and message).	
:ENABle <list>	Specify error and status messages for error queue.	Note 2
:ENABle?	Read the enabled messages.	
:DISable <list>	Specify messages not to be placed in queue.	Note 2
:DISable?	Read the disabled messages.	
:CLEar	Clear messages from error queue.	
SYSTem	SYSTem subsystem:	
:ERRor	Read error queue:	Note 1
[:NEXT]?	Read and clear oldest error and status (code and message).	
:ALL?	Read and clear all errors and status (code and message).	
:COUNt?	Read the number of messages in queue.	
:CODE	Code numbers only:	
[:NEXT]?	Read and clear oldest error and status (code only).	
:ALL?	Read and clear all errors and status (codes only).	
:CLEar	Clear messages from error queue.	

Notes:
1. Power-up and *CLS empty the error queue. STATus:PRESet has no effect.
2. Power-up enables error messages and disables status messages. *CLS and STATus:PRESet have no effect.

Programming example: Read error queue

The following command reads the error queue:

STAT:QUE?

Common Commands

In this section:

Command summary

Common commands (summarized in Table 14-1) are device commands that are common to all devices on the bus. These commands are designated and defined by the IEEE Std 488.2 standard. Most of these commands are described in detail in this section.

NOTE   *The following common commands associated with the status structure are covered in Section 13: *CLS, *ESE, *ESE?, *ESR?, *SRE, *SRE?, and *STB?.*

Table 14-1
IEEE-488.2 common commands and queries

Mnemonic	Name	Description
*CLS	Clear status	Clears all event registers and the error queue.
*ESE <NRf>	Event enable command	Program the standard event enable register.
*ESE?	Event enable query	Read the standard event enable register.
*ESR?	Event status register query	Read and clear the standard event enable register.
*IDN?	Identification query	Returns the manufacturer, model number, serial number, and firmware revision levels of the instrument.
*OPC	Operation complete command	Set the operation complete bit in the standard event register after all pending commands have been executed.
*OPC?	Operation complete query	Places an ASCII "1" into the output queue when all pending selected device operations have been completed.
*OPT?	Option query	Queries the Model 6482 for a list of installed options.
*RCL <NRf>	Recall command	Returns the Model 6482 to the user-saved setup.
*RST	Reset command	Returns the Model 6482 to the *RST default conditions.
*SAV <NRf>	Save command	Saves the present setup as the user-saved setup.
*SRE <NRf>	Service request enable command	Programs the service request enable register.
*SRE?	Service request enable query	Reads the service request enable register.
*STB?	Status byte query	Reads the status byte register.
*TRG	Trigger command	Sends a bus trigger to the Model 6482.
*TST?	Self-test query	Performs a checksum test on ROM and returns the result.
*WAI	Wait-to-continue command	Waits until all previous commands are executed.

Command reference

*IDN? — Identification query

This command reads the identification code. The identification code includes the manufacturer, model number, serial number, and firmware revision levels and is sent in the following format:

KEITHLEY INSTRUMENTS INC., Model 6482, xxxxxxx, yyyyy/zzzzz /a/d

Where: xxxxxxx = the serial number.
yyyyy/zzzzz =s the firmware revision levels of the digital board ROM and display board ROM, including date and time of build.
a = the analog board revision level.
d = the digital board revision level.

*OPC — Operation complete

This command sets the OPC bit.

*OPC? — Operation complete query

This command places a "1" in output queue. When *OPC is sent, the OPC bit in the standard event register will set after all pending command operations are complete. When *OPC? is sent, an ASCII "1" is placed in the output queue after all pending command operations are complete.

Typically, either one of these commands is sent after the INITiate command. The INITiate command is used to take the instrument out of the idle state so it can perform measurements. While operating within the trigger model layers, all sent commands (except DCL, SDC, IFC, SYSTem:PRESet, *RST, *RCL, *TRG, GET, and ABORt) will not execute.

After all programmed operations are completed, the instrument returns to the idle state and then executes all pending commands (including *OPC and *OPC?). After the last pending command is executed, the OPC bit and/or an ASCII "1" is placed in the output queue.

*OPC programming example

The command sequence in Table 14-2 will perform 10 measurements. After the measurements are completed (in approximately 10 seconds), an ASCII "1" will be placed in the output queue and displayed on the computer. Note that additional codes must be added to query the instrument for the presence of the ASCII "1" in the output queue.

Table 14-2
*OPC programming example

Command	Description

`*RST`
`:TRIG:DEL 1`
`:ARM:COUN 10`
`:OUTP1 ON`
`:INIT`
`*OPC?`
	Return the Model 6482 to GPIB defaults (idle).
	Set trigger delay for 1 second.
	Program for to take 10 measurements and then stop.
	Turn on the output.
	Start taking measurements.
	Send *OPC? to query the output queue. Additional code required to test for "1" in output queue.

*OPT? — Option query

This command returns a list of installed options. When *OPT? is sent, the Model 6482 returns a list of any installed options.

*SAV <NRf> — Save setup

This command saves the present setup in memory.

Parameters   0 = Memory location 0
1 = Memory location 1
2 = Memory location 2
3 = Memory location 3
4 = Memory location 4

Use the *SAV command to save the present instrument setup configuration in memory for later recall. Any control affected by *RST can be saved by the *SAV command. See *RCL <NRf> — Recall setup for additional details.

*RCL <NRf> — Recall setup

The *RCL command is used to restore the instrument to a saved setup configuration. Five setup configurations can be saved and recalled.

The Model 6482 ships from the factory with SYSTem:PRESet defaults loaded into the available setup memory. If a recall error occurs, the setup memory defaults to the SYSTem:PRESet values.

*SAV, *RCL programming example

Table 14-3 summarizes the basic command sequence for saving and recalling a setup. The present setup is stored in memory location 2, GPIB defaults are restored, and the memory location 2 setup is recalled.

Table 14-3
*SAV, *RCL programming example

Command	Description
*SAV 2	Save present setup in memory location 2.
*RST	Restore GPIB defaults.
*RCL 2	Recall location 2 setup.

*RST — Reset

This command returns the Model 6482 to GPIB defaults. When the *RST command is sent, the Model 6482 performs the following operations:

- Returns the Model 6482 to the GPIB default conditions. Refer to the "Default parameters" column of SCPI tables in Section 16.
- Cancels all pending commands.
- Cancels response to any previously received *OPC and *OPC? commands.

*TRG — Trigger

This command sends a bus trigger to Model 6482. Use the *TRG command to issue a GPIB trigger to the Model 6482. It has the same effect as a group execute trigger (GET).

Use the *TRG command as an event to control operation. The Model 6482 reacts to this trigger if BUS is the programmed arm control source. The control source is programmed from the TRIGger subsystem.

NOTE   *Details on triggering are covered in Section 8.*

*TRG programming example

The command sequence in Table 14-4 configures the Model 6482 to be controlled by bus triggers. The last command, which sends a bus trigger, triggers one measurement. Each subsequent bus trigger will also trigger a single measurement.

NOTE   *With :ARM:SOURce BUS selected, do not send any commands (except *TRG, GET, DCL, SDC, IFC, and ABORt) while performing source-measure operations. If you do, erratic operation will occur.*

Table 14-4
*TRG programming example

Command	Description
*RST	Restore GPIB defaults.
:ARM:SOUR BUS	Select BUS control source.
:ARM:COUN INF	Set arm layer count to infinite.
:OUTP1 ON	Turn on output.
:INIT	Take Model 6482 out of the idle state.
*TRG	Trigger one measurement.

*TST? — Self-test query

This command runs a self test and reads the result. Use this query command to perform a checksum test on ROM. The command places the coded result (0 or 1) in the output queue. When the Model 6482 is addressed to talk, the coded result is sent from the output queue to the computer.

A returned value of zero (0) indicates that the test passed, and a value of one (1) indicates that the test failed.

*WAI — Wait-to-continue

Effectively, the *WAI command is a No-Op (no operation) for the Model 6482 and thus, does not need to be used.

The *WAI command is used to suspend the execution of subsequent commands until the device operations of all previous overlapped commands are finished. The *WAI command is not needed for sequential commands.

Two types of device commands exist:

- **Sequential commands**: A command whose operations need to finish before the next command is executed.
- **Overlapped commands**: A command that allows the execution of subsequent commands while device operations of the overlapped command are still in progress.

This page left blank intentionally.

SCPI Measurement Commands

In this section:

Command summary

The measurement commands are used to acquire readings. You can use these high-level instructions to control the measurement process. These commands are summarized in Table 15-1.

Table 15-1
Signal-oriented measurement command summary

Command	Description

`:CONFigure:CURRent[:DC]`
`:CONFigure?`
`:FETCh?`
`:READ?`
`:MEASure[:CURRent[:DC]]?`
	Configures Model 6482 for current measurements. Output turns on.
	Returns active function (amperes).
	Requests latest readings.
	Performs an :INITiate and a :FETCh? operation.
	Single measurement mode. Performs one :CONFigure and one :READ? operation.

Configuring measurement function

This topic provides detailed information on commands that can configure the measurement function.

CONFigure:CURRent[:DC]

Query   :CONFigure?   Returns active function (current).

Description   This command configures the instrument to measure DC current. The :READ? command is then typically used to trigger a specified number of measurements. See "READ?," page Section 15-4.

When this command is sent, the Model 6482 will be configured as follows:

- Select current function.
- All controls related to the current function are defaulted to the *RST values.
- The event control source of the trigger model is set to Immediate.
- The count values of the trigger model are set to one.
- The delay of the trigger model is set to zero (0).
- All math calculations are disabled.
- Buffer operation is disabled.
- Autozero is enabled.
- Both source outputs will turn on.

WARNING   ***When :CONFigure is sent, the outputs will turn on. Beware of hazardous voltage that may be present on the output terminals.***

NOTE   *This command is automatically asserted when the :MEASure? command is sent.*

Acquiring readings

This topic describes commands to acquire post-processed readings, both trigger and acquire readings, and to perform a single measurement.

FETCh?

Description   This query command requests the latest post-processed readings stored in the sample buffer. After sending this command and addressing the Model 6482 to talk, the readings are sent to the computer. This command does not affect the instrument setup.

This command does not trigger source and measure operations; it requests the last available readings. Note that this command can repeatedly return the same readings. Until there are new readings, this command continues to return the old readings. For example, assume that the Model 6482 performed 20 source and measure operations. The :FETCh? command will request the readings for those 20 source and measure operations. If :FETCh? is sent while performing source and measure operations (ARM indicator on), it will not be executed until the Model 6482 goes back into an idle state.

NOTE   *The :FETCh? command is automatically asserted when the :READ? or :MEASure? command is sent.*

The readings that are acquired depend on which data elements are selected (Section 16, "FORMat subsystem," "Data elements"). Measure readings take priority over source readings, and data from channels not sourced or measured are assigned the NAN (not a number) value of +9.91e37.

NOTE   *Send :FORM:ELEM CURR1 before :FETCh? for channel 1 readings or :FORM:ELEM CURR2 for channel 2 readings.*

See Appendix B for a detailed explanation on how data flows through the various operation blocks of the Model 6482. It clarifies the types of readings that are acquired by the commands to read data.

DATA[:LATest]?

Description   This command works exactly like FETCh?, except that it returns only the most recent reading from the channel selected by :FORM:ELEM.

READ?

Description   This command is used to trigger and acquire readings. The number of readings depends on how the trigger model is configured. For example, if configured for 20 source and measure operations (arm count 1, trigger count 20), then 20 sets of readings will be acquired after the Model 6482 returns to the idle state.

When this command is sent, the following commands execute in the order that they are presented:

- :INITiate
- :FETCh?

The :INITiate command starts operation by taking the instrument out of an idle state.

NOTE   *:READ? is illegal when the :ARM:SOUR BUS command is in effect. See Section 8, "Triggering," for details.*

After all source and measure operations are completed, the Model 6482 goes back into an idle state, and then the :FETCh? command is executed. The readings are sent to the computer and displayed when the Model 6482 is addressed to talk.

NOTE   *The :FORM:ELEM command controls the reading source. To access channel 1 readings, send :FORM:ELEM CURR1 before sending :READ?. To access channel 2 readings, send :FORM:ELEM CURR2 before using :READ?. See Section 16, "FORMat subsystem," for details.*

READ? will work with the output off. If in the sweep mode with a trigger count >1, the source value on the display will update; however, the actual output will be 0 V.

If automatic output-off is enabled (:SOURce1:CLEar:AUTO ON or :SOURce2:CLEar:AUTO ON), then the corresponding output will automatically turn on at the beginning of each SDM (source-delay-measure) cycle and turn off after each measurement.

NOTE   *See Appendix B for a detailed explanation on how data flows through the various operation blocks of the* Model 6482. *It clarifies the type of readings that are acquired by the various commands to read data.*

MEASure[:CURRent[:DC]]?

Parameters   CURRent[:DC]   Amps function

Description   This command combines other signal-oriented measurement commands to perform a single measurement and acquire the reading.

When this command is sent, the following commands execute in the order that they are presented.

- :CONFigure:CURRent[:DC]
- :READ?

When :CONFigure is executed, the instrument goes into a single measurement mode. See "CONFigure:CURRent[:DC]" for more details.

NOTE   *:MEAS? is illegal when the :ARM:SOUR BUS command is in effect. See Section 8, "Triggering," for details.*

When :READ? is executed, its operations will then be performed. In general, another :ABORt is performed, then an :INITiate, and finally a FETCh? to acquire the readings. See "READ?," page Section 15-4, for more details.

NOTE   *The :FORM:ELEM command controls the reading source. To access channel 1 readings, send :FORM:ELEM CURR1 before sending :MEAS?. To access channel 2 readings, send :FORM:ELEM CURR2 before using :MEAS?.*

When :MEASure? is sent, the source turns on and a single measurement is performed. If automatic output-off is enabled (:SOURce1:CLEar:AUTO ON or :SOURce2:CLEar:AUTO ON), then the corresponding output will turn off after the measurement is completed. If automatic output-off is disabled (:SOURce1:CLEar:AUTO OFF or :SOURce2:CLEar:AUTO OFF), then the corresponding output will remain on after the measurement is completed.

WARNING   ***With automatic output-off disabled, the output will remain on after the single source-measure operation is performed. Beware of hazardous voltage that may be present on the output terminals.***

This page left blank intentionally.

SCPI Command Reference

In this section:

Reference tables

Table 16-1 through Table 16-10 summarize the commands for each SCPI subsystem. The following list includes the SCPI subsystem commands and the table number where each command is summarized.

Summary table	Subsystem	Function
Table 16-1	CALCulate	Math, relative offset, ratio, delta, limits, buffer statistics.
Table 16-2	DISPlay	Display format and messages.
Table 16-3	FORMat	Remote data format.
Table 16-4	OUTPut	Source 1 and source 2 output on-off, output enable.
Table 16-5	SENSe	Channel 1 and channel 2 current measurements.
Table 16-6	SOURce	Channel 1 and channel 2 source configuration.
Table 16-7	STATus	Instrument status.
Table 16-8	SYSTem	System presets, error formats, automatic zero, line frequency.
Table 16-9	TRACe	Channel 1 and channel 2 buffer control and formats.
Table 16-10	TRIGger	Instrument triggering control.

General notes:

Brackets ([ ]) are used to denote optional character sets. These optional characters do not have to be included in the program message. Do not use brackets in the program message.

Angle brackets (< >) are used to indicate parameter type. Do not use angle brackets in the program message.

The Boolean parameter (<b>) is used to enable or disable an instrument operation. One (1) or ON enables the operation; zero (0) or OFF disables the operation.

Upper case characters indicate the short-form version for each command word.

Default parameter: Listed parameters are both the *RST and :SYSTem:PRESet defaults, unless noted differently. Parameter notes are located at the end of each table.

SCPI: A check mark (✓) indicates that the command and its parameters are SCPI confirmed. An unmarked command indicates that it is a SCPI command, but does not conform to the SCPI standard set of commands. It is not a recognized command by the SCPI consortium. SCPI confirmed commands that use one or more non-SCPI parameters are explained by notes.

Table 16-1
CALCulate command summary

Command	Description	Default parameter	SCPI
`:CALCulate[1]`	Subsystem to control CALC1 (channel 1) math:		✓
` :FORMat <name>`	Select math format (MXB[1], COND[1], POWER[1], or RES[1]).	MXB[1]	✓
` :KMATh`	Configure MX + B parameters.		
` :MBFactor <n>`	Set B parameter (-9.99999e20 to 9.99999e20).	0	
` :MBFactor?`	Query B parameter.		
` :MMFactor <n>`	Set M parameter (-9.99999e20 to 9.99999e20).	1	
` :MMFactor?`	Query M parameter.		
` :MUNits "<string>"`	Set MX + B units (1 to 3-character ASCII string).	"MXB"	
` :MUNits?`	Query M parameter.		
` :STATe <b>`	Enable or disable CALC1 math.	OFF	✓
` :STATe?`	Query state of math.		✓
` :DATA`	Path to CALC1 data.		
` :LATest?`	Return only most recent math result.		
` :DATA?`	Read result of math.		✓
`:CALCulate2`	Subsystem to control CALC2 (channel 2) math:		✓
` :FORMat <name>`	Select math format (MXB2, COND2, POWER2, or RES2).	MXB2	✓
` :KMATh`	Configure MX + B parameters.		
` :MBFactor <n>`	Set B parameter (-9.99999e20 to 9.99999e20).	0	
` :MBFactor?`	Query B parameter.		
` :MMFactor <n>`	Set M parameter (-9.99999e20 to 9.99999e20).	1	
` :MMFactor?`	Query M parameter.		
` :MUNits "<string>"`	Set MX + B units (1 to 3-character ASCII string).	"MXB"	
` :MUNits?`	Query M parameter.		
` :STATe <b>`	Enable or disable CALC2 math.	OFF	
` :STATe?`	Query state of math.		✓
` :DATA`	Path to CALC2 data.		✓
` :LATest?`	Return only most recent math result.		
` :DATA?`	Read result of math.		✓
`:CALCulate3`	Subsystem to control CALC3 (channel 1 RELATIVE OFFSET):		✓
` :FEED <name>`	Select input path (SENSe[1] or CALCulate[1]).	SENS1	✓
` :FEED?`	Query CALC3 feed.		✓
` :NULL`	Path to configure and control REL:		
` :OFFSet <NRf>`	Specify REL value (-9.999999e20 to 9.999999e20).	0	
` :OFFSet?`	Query REL value.		
` :STATe <b>`	Enable or disable REL.	OFF	
` :STATe?`	Query state of REL.		
` :ACQuire`	Automatically acquire REL value.		
` :DATA?`	Read CALC3 data.		✓
`:CALCulate4`	Subsystem to control CALC4 (channel 2 REL):		✓
` :FEED <name>`	Select input path (SENSe2 or CALCulate2).	SENS2	✓
` :FEED?`	Query CALC4 feed.		✓
` :NULL`	Path to configure and control REL:		
` :OFFSet <NRf>`	Specify REL value (-9.999999e20 to 9.999999e20).	0	
` :OFFSet?`	Query REL value.		
` :STATe <b>`	Enable or disable REL.	OFF	
` :STATe?`	Query state of REL.		
` :ACQuire`	Automatically acquire REL value.		
` :DATA?`	Read CALC4 data.		✓

Table 16-1 (cont.)
CALCulate command summary

Command	Description	Default parameter	SCPI
:CALCulate5	Subsystem to control CALC5 (RATIO):		✓
:FORMat <name>	Select ratio math; C3C4 (CALC3/CALC4); C4C3 (CALC4/CALC3).	C3C4	✓
:STATe <b>	Enable or disable CALC5 math.	OFF	
:STATe?	Query state of ratio.		
:DATA?	Request CALC5 data.		✓
:CALCulate6	Subsystem to control CALC6 (DELTA):		✓
:FORMat <name>	Select delta math; C3C4 (CALC3 - CALC4); C4C3 (CALC4 - CALC3).	C3C4	✓
:STATe <b>	Enable or disable CALC5 math.	OFF	
:STATe?	Query state of delta.		
:DATA?	Request to CALC5 data.		✓
:CALCulate7	Path to CALC7 (limit tests):		
:DATA?	Return all CALC7 results.		
:LATest?	Return most recent CALC7 result.		
:FEED <name>	Specify Limit 3 to 6 input path (CALCulate3, CALCulate4, CALCulate5, or CALCulate6).		
:FEED?	Query input path.		
:LIMit[1]	Path to channel 1 hardware (compliance limits).	IN	✓
:COMPliance	Path to mode.	15 or 7	✓
:FAIL <name>	Set fail mode (IN or OUT) of compliance		✓
:FAIL?	Query fail mode		
:STATe <b>	Enable or disable Limit 1 test.		
:STATe?	Query state of Limit 1 test.		
:FAIL?	Returns result of Limit 1 test: 0 (pass) or 1 (fail).		
:LIMit2	Path to channel 2 hardware (compliance) limit.		
:COMPliance	Path to mode.		
:FAIL <name>	Set fail mode (IN or OUT) of compliance	IN	
:FAIL?	Query fail mode	15 or 7	
:STATe <b>	Enable or disable Limit 2 test.		✓
:STATe?	Query state of Limit 2 test.	OFF	✓
:FAIL?	Returns result of Limit 2 test: 0 (pass) or 1 (fail).		✓
:LIMitX	Path to software limits. LIMitX = LIMit3...LIMit6.[3]		✓
:UPPer	Configure upper limit:		✓
[:DATA] <n>	Specify upper limit (-9.999999e20 to 9.999999e20).	1	✓
[:DATA]?	Query upper limit.		✓
:LOWer	Configure lower limit:		✓
[:DATA] <n>	Specify lower limit (-9.999999e20 to 9.999999e20).	-1	✓
[:DATA]?	Query lower limit.		✓
:PASS	Path to sorting mode pass:		
:STATe <b>	Enable or disable Limit X test.	OFF	✓
:STATe?	Query state of Limit X test.		✓
:FAIL?	Return result of Limit X test: 0 (pass) or 1 (fail).		✓
:CLIMits	Composite limits for Limit 1 through Limit 6		

Table 16-1 (cont.)
CALCulate command summary

Command	Description	Default parameter	SCPI
:CLEar	Clear test results:		
[:IMMediate]	Clear latest limit test result.		
:AUTO \<b>	Enable or disable clearing of test results when :INITiate command is sent.	ON	
:AUTO?	Query state of automatic clear.		
:CALCulate8	Subsystem to control CALC8 (channel 1 buffer stats):		✓
:FORMat \<name>	Specify math format (MEAN, SDEViation, MAXimum, MINimum, or PKPK).	MEAN	✓
:FORMat?	Query math format.		✓
:DATA?	Read math result of CALC8.		✓

[3] LIMitX = LIMit3, LIMit4, LIMit5, or LIMit6.

Table 16-2
DISPlay command summary

Command	Description	Default parameter	SCPI
:DISPlay			
:ENABle \<b>	Turn on or turn off front panel display.	Note 1	✓
:ENABle?	Query state of display.		✓
:MODE \<name>	Select display function. Name = CALC3, CALC4, CALC5, CALC6, or DUAL.	DUAL	
:MODE?	Query selected display function.		
[:WINDow[1]]	Path to locate message to bottom display:		✓
:TEXT	Control user test message:	Note 2	✓
:DATA \<a>	Define ASCII message "a" (up to 32 characters).		✓
:DATA?	Query text message.		✓
:STATe \<b>	Enable or disable message mode.	Note 3	✓
:STATe?	Query text message state.		✓
:DATA?	Query data on bottom portion of display.		
:ATTRibutes?	Query attributes of message characters: blinking (1) or not blinking (0).		
:WINDow2	Path to locate message to bottom display:		
:TEXT	Control user test message:	Note 2	✓
:DATA \<a>	Define ASCII message "a" (up to 32 characters).		✓
:DATA?	Query text message.		✓
:STATe \<b>	Enable or disable message mode.	Note 3	✓
:STATe?	Query text message state.		✓
:DATA?	Query data on bottom portion of display.		✓
:ATTRibutes?	Query attributes of message characters: blinking (1) or not blinking (0).		
:DIGits\<n>	Specify display resolution (4 to 7).	6	
:DIGits?	Query display resolution.		

Notes:

1. *RST and :SYSTem:PRESet has no effect on the display circuitry. Pressing LOCAL or cycling power enables (ON) the display circuit.
2. *RST and :SYSTem:PRESet has no effect on a user-defined message. Pressing LOCAL or cycling power cancels all user-defined messages.
3. *RST and :SYSTem:PRESet has no effect on the state of the message mode. Pressing LOCAL or cycling power disables (OFF) the message mode.

Table 16-3
FORMat command summary

Command	Description	Default parameter	SCPI
:FORMat			
:SREGister <name>	Select data format for reading status event registers (ASCii, HEXadecimal, OCTal or BINary).	ASCii	✓
:SREGister?	Query format for reading status event registers.		
[:DATA] <type>[<,length>]	Specify data format (ASCii, REAL, 32 or SREal).	ASCii	✓
[:DATA]?	Query data format.		✓
:BORDer <name>	Specify byte order (NORMal or SWAPped).	Note	✓
:BORDer?	Query byte order.		✓
:ELEMents			
[:SENSe[1]] <item list>	Specify data elements (CURRent[1], CURRent2, TIME, and STATus).	CURR1, CURR2	
[:SENSe[1]]?	Query data format elements.		
:CALCulate <item list>	Specify CALCulate elements (CALCulate, TIME, or STATus).*	CALC	
:CALCulate?	Query CALC data elements,		
:TRACe <item list>	Specify TRACe data elements: CURRent[1], CURRent2, CALCulate1, CALCulate2, CALCulate3, CALCulate4, CALCulate5, CALCulate6, CALCulate7, TIME, STATus, ALL, DEFault (CURRent[1], CURRent2).	DEFault	
:TRACe?	Query TRACe elements.		
:SOURce3 <name>	Specify SOURce3 data format (ASCii, HEXadecimal, OCTal, or BIN).	ASCii	
:SOURce3?	Query SOURce3 data format.		

Note: *RST default is NORMal. :SYST:PRES default is SWAPPed.
*This command specifies data elements for CALC1 through CALC7.

Table 16-4
OUTPut command summary

Command	Description	Default parameter	SCPI
:OUTPut[1]	Path to control output 1.		✓
[:STATe] <b>	Turn source 1 on or off.	OFF	✓
[:STATe]?	Query state of source 1.		✓
:ENABle	Path to control output enable:		
[:STATe] <b>	Enable or disable output enable for both sources.	OFF	
[:STATe]?	Query state of output enable.		
:TRIPped?	Output enable tripped?: 1 (no) or 0 (yes).		
:OUTPut2	Path to control output 2.		✓
[:STATe] <b>	Turn source 2 on or off.	OFF	✓
[:STATe]?	Query state of source 2.		✓

Table 16-5
SENSe command summary

Command	Description	Default parameter	SCPI
[:SENSe[1]]	Sense 1 subsystem to control channel 1 measurement:		✓
:CURRent[:DC]	Path to configure current:		✓
:RANGe	Configure measurement range:		✓
[:UPPer] <n>	Select range by specifying the expected current reading; 0 to ±21e-3.	2e-4	✓
[:UPPer]?	Query range.		✓
:AUTO <b>	Enable or disable autorange.	ON	✓
:AUTO?	Query autorange.		✓
:ULIMit <n>	Set autoranging upper limit (-0.021 to 0.021).	2e-2	✓
:ULIMit?	Query autoranging upper limit.		✓
:LLIMit <n>	Set autoranging lower limit (-0.021 to 0.021).	2e-9	✓
:LLIMit?	Query autoranging lower limit.		✓
:NPLCycles <n>	Specify integration rate (in line cycles): 0.01 to 10.	1	✓
:NPLCycles?	Query integration rate.		✓
:AVERage	Path to configure moving and repeat filters:		✓
:COUNt <n>	Specify filter count; 1 to 100.	10	✓
:COUNt?	Query filter count.		✓
[:STATe] <b>	Enable or disable digital filter.	OFF	✓
[:STATe]?	Query state of digital filter.		✓
:TCONtrol <name>	Select filter control (MOVing or REPeat).	REPeat	
:TCONtrol?	Query filter control mode.		
:ADVanced	Configure the advanced filter:		✓
:NTOLerance <NRf>	Set filter noise window (in %); 0 to 105.	5	✓
:NTOLerance?	Query filter noise window setting.		✓
[:STATe] <b>	Enable or disable advanced filter.	OFF	✓
[:STATe]?	Query state of advanced filter.		✓
:MEDian	Configure and control the median filter:		
:RANK <NRf>	Specify median filter rank; 1 to 5.	1	
:RANK?	Query median filter rank.		
[:STATe] <b>	Enable or disable median filter.	OFF	
[:STATe]?	Query state of median filter.		

Table 16-5 (cont.)
SENSe command summary

Command	Description	Default parameter	SCPI
:SENSe2	Sense 2 subsystem to control channel 2 measurement:		✓
:CURRent[:DC]	Path to configure current:		✓
:RANGe	Configure measurement range:		✓
[:UPPer] <n>	Select range by specifying the expected current reading; 0 to ±21e-3.	2e-4	✓
[:UPPer]?	Query range.		✓
:AUTO <b>	Enable or disable autorange.	ON	✓
:AUTO?	Query autorange.		✓
:ULIMit <n>	Set autoranging upper limit (-0.021 to 0.021).	2e-2	✓
:ULIMit?	Query autoranging upper limit		✓
:LLIMit <n>	Set autoranging lower limit (-0.021 to 0.021).	2e-9	✓
:LLIMit?	Query autoranging lower limit.		✓
:NPLCycles <n>	Specify integration rate (in line cycles): 0.01 to 10.	1	✓
:NPLCycles?	Query integration rate.		✓
:AVERage	Path to configure moving and repeat filters:		✓
:COUNt <n>	Specify filter count; 1 to 100.	10	✓
:COUNt?	Query filter count.		✓
[:STATe] <b>	Enable or disable digital filter.	OFF	✓
[:STATe]?	Query state of digital filter.		✓
:TCONtrol <name>	Select filter control (MOVing or REPeat).	REPeat	
:TCONtrol?	Query filter control mode.		
:ADVanced	Configure the advanced filter:		✓
:NTOLerance <NRf>	Set filter noise window (in %); 0 to 105.	5	✓
:NTOLerance?	Query filter noise window setting.		✓
[:STATe] <b>	Enable or disable advanced filter.	OFF	✓
[:STATe]?	Query state of advanced filter.		✓
:MEDian	Configure and control the median filter:		
:RANK <NRf>	Specify median filter rank; 1 to 5.	1	
:RANK?	Query median filter rank.		
[:STATe] <b>	Enable or disable median filter.	OFF	
[:STATe]?	Query state of median filter.		

Table 16-6
SOURce command summary

Command	Description	Default parameter	SCPI
:SOURce[1]	Path to control channel 1 source:		✓
:CLEar	Path to clear source:		
[:IMMediate]	Turn selected source off.		
:AUTO	Automatically turn source on/off.	OFF	
:MODE <name>	Specify auto on/off mode (ALWays or TCOunt).	ALWays	
:MODE?	Query auto on/off mode.		
:DELay <n>	Specify settling time (in sec): 0 to 9999.999.	0.001	
:AUTO <b>	Enable or disable automatic settling time.		
:AUTO?	Query state of automatic settling time.		
:DELay?	Query source settling time.		
:GCONnect <b>	Enable or disable ground connect mode.	OFF	
:GCONnect?	Query ground connect state.		
:VOLTage	Path to configure V-Source:		✓
:MODE <n>	Select V-Source mode (FIXed, SWEep, or LIST).	FIXed	✓
:MODE?	Query V-Source mode.		✓
:RANGe <n>	Select fixed V-Source range; -30 to 30 (10 or 30).	10	✓
:AUTO <b>	Enable or disable autoranging.	ON	✓
:AUTO?	Query state of autoranging.		✓
:RANGe?	Query V-Source range setting.		✓
[:LEVel]	Set V-Source level (in volts):		✓
[:IMMediate]	Set specified voltage level immediately:		✓
[:AMPLitude] <n>	Specify voltage level; -30 to 30.	0	✓
[:AMPLitude]?	Query voltage level.		✓
:TRIGgered	Set specified voltage level when triggered:		✓
[:AMPLitude] <n>	Specify voltage level; -30 to 30.	0	✓
[:AMPLitude]?	Query voltage level.		✓

Table 16-6 (cont.)
SOURce command summary

Command	Description	Default parameter	SCPI
:SOURce[1]	Path to configure channel 1 source (continued):		
:VOLTage			
:STARt <n>	Specify start level for V-sweep; -30 to 30.	0	✓
:STARt?	Query start level for voltage sweep.		✓
:STOP <n>	Specify stop level for V-sweep; -30 to 30.	0	✓
:STOP?	Query stop level for voltage sweep.		✓
:STEP <n>	Specify step value for V-sweep; -30 to 30.	0	
:STEP?	Query step value for voltage sweep.		
:SPAN <n>	Specify span; -30 to 30.	0	✓
:SPAN?	Query span.		✓
:CENTer <n>	Specify center point; -30 to 30.	0	✓
:CENTer?	Query center point.		✓
:SWEep	Configure SWEep source mode:		✓
:SPACing <name>	Select sweep spacing type (LINear or LOGarithmic).	LINear	✓
:SPACing?	Query sweep spacing.		✓
:POINts <n>	Specify number of sweep points (2 to 3000).	3000	
:POINts?	Query number of points in sweep.		
:DIRection <name>	Sweep from start to stop (UP) or from stop to start (DOWN).	UP	
:DIRection?	Query sweep direction.		
:RANGing <name>	Select source ranging mode (BEST, AUTO, or FIXed).	BEST	
:RANGing?	Query source ranging mode.		
:LIST	Configure LIST source mode:	No effect	
:VOLTage <NRf>	Create list of V-Source values; -30 to 30.		
:APPend <NRf>	Add up to 30 V-Source values to end of list.		
:POINts?	Query number of source values in list.		
:VOLTage?	Query V-Source list.		

Table 16-6 (cont.)
SOURce command summary

Command	Description	Default parameter	SCPI
:SOURce2	Path to control channel 2 source:		✓
:CLEar	Path to clear source:		
[:IMMediate]	Turn selected source off.		
:AUTO	Select source auto on/off mode source.		
:MODE <name>	Specify auto on/off mode (ALWays or TCOunt).	ALWays	
:MODE?	Query auto on/off mode.		
:DELay <n>	Specify settling time (in sec): 0 to 9999.999.	0.001	
:AUTO <b>	Enable or disable auto settling time.	ON	
:AUTO?	Query state of auto settling time.		
:DELay?	Query source settling time.		
:GCONnect <b>	Enable or disable ground connect mode.	OFF	
:GCONnect?	Query ground connect state.		
:VOLTage	Path to configure V-Source:		✓
:MODE <n>	Select V-Source mode (FIXed, SWEep, or LIST).	FIXed	✓
:MODE?	Query V-Source mode.		✓
:RANGe <n>	Select fixed V-Source range; -30 to 30 (10 or 30).	10	✓
:AUTO <b>	Enable or disable autoranging.	ON	✓
:AUTO?	Query state of autoranging.		✓
:RANGe?	Query V-Source range setting.		✓
[:LEVel]	Set V-Source level (in volts):		✓
[:IMMediate]	Set specified voltage level immediately:		✓
[:AMPLitude] <n>	Specify voltage level; -30 to 30.	0	✓
[:AMPLitude]?	Query voltage level.		✓
	Set specified voltage level when triggered:		✓
:TRIGgered	Specify voltage level; -30 to 30.	0	✓
[:AMPLitude] <n>	Query voltage level.		✓
[:AMPLitude]?			

Table 16-6 (cont.)
SOURce command summary

Command	Description	Default parameter	SCPI
:SOURce2	Path to configure channel 2 source (continued):		
:VOLTage			
:STARt <n>	Specify start level for V-sweep; -30 to 30.	0	✓
:STARt?	Query start level for voltage sweep.		✓
:STOP <n>	Specify stop level for V-sweep; -30 to 30.	0	✓
:STOP?	Query stop level for voltage sweep.		✓
:STEP <n>	Specify step value for V-sweep; -30 to 30.	0	
:STEP?	Query step value for voltage sweep.		
:SPAN <n>	Specify span; -30 to 30.	0	✓
:SPAN?	Query span.		✓
:CENTer <n>	Specify center point; -30 to 30.	0	✓
:CENTer?	Query center point.		✓
:SWEep	Configure SWEep source mode:		✓
:SPACing <name>	Select sweep spacing type (LINear or LOGarithmic).	LINear	✓
:SPACing?	Query sweep spacing.		✓
:POINts <n>	Specify number of sweep points (2 to 3000).	3000	
:POINts?	Query number of points in sweep.		
:DIRection <name>	Sweep from start to stop (UP) or from stop to start (DOWN).	UP	
:DIRection?	Query sweep direction.		
:RANGing <name>	Select source ranging mode (BEST, AUTO, or FIXed).	BEST	
:RANGing?	Query source ranging mode.		
:LIST	Configure LIST source mode:	No effect	
:VOLTage <NRf>	Create list of V-Source values; -30 to 30.		
:APPend <NRf>	Add up to 30 V-Source values to end of list.		
:POINts?	Query number of source values in list.		
:VOLTage?	Query V-Source list.		

Table 16-7
STATus command summary

Command	Description	Default parameter	SCPI
:STATus		Note 1	✓
:MEASurement	Control measurement event registers:		
[:EVENt]?	Read the event register.[6]	Note 2	✓
:ENABle <NDN>	Program the enable register.	Note 3	✓
or <NRf>			
:ENABle?	Read the enable register.[6]		✓
:CONDition?	Read the condition register.[6]		✓
:OPERation	Control operation status registers:		✓
[:EVENt]?	Read the event register.[6]	Note 2	✓
:ENABle <NDN>	Program the enable register.	Note 3	✓
or <NRf>			
:ENABle?	Read the enable register.[6]		✓
:CONDition?	Read the condition register.[6]		✓
:QUEStionable	Control questionable status registers:		✓
[:EVENt]?	Read the event register.[6]	Note 2	✓
:ENABle <NDN>	Program the enable register.	Note 3	✓
or <NRf>			
:ENABle?	Read the enable register.[6]		✓
:CONDition?	Read the condition register.[6]		✓
:PRESet	Return status registers to default states.		✓
:QUEue	Path to access error queue:		
[:NEXT]?	Read the most recent error message.	Note 4	✓
:ENABle <list>	Specify error and status messages for error queue.	Note 5	✓
:ENABle?	Read the enabled messages.		✓
:DISable <list>	Specify messages not to be placed in error queue.	Note 5	
:DISable?	Read the disabled messages.		
:CLEar	Clears all messages from error queue.		

Notes:

1. Commands in this subsystem are not affected by *RST and :SYSTem:PREset. The effects of cycling power, *CLS and :STATus:PRESet, are explained by the following notes.

2. Event registers: Power-up and *CLS clears all bits. :STATus:PRESet has no effect.

3. Enable registers: Power-up and :STATus:PRESet clears all bits. *CLS has no effect. Accepts the SCPI 1995.0 mandated (nondecimal numeric) format (#H, #Q, or #B).

4. Error queue: Power-up and *CLS clears all bits of the registers.

5. Error queue messages: Power-up clears list of messages. *CLS and :STATus:PRESet have no effect.

6. Register query commands: The format for the response messages (ASCII, hexadecimal, octal, or binary) depends on which data format is presently selected. See the :FORMat:SREGister command.

Table 16-8
SYSTem command summary

Command	Description	Default parameter	SCPI
:SYSTem			
:PRESet	Return to :SYSTem:PRESet defaults.		✓
:POSetup <name>	Select power-on setup (RST, PRESet or SAV 0-4).		
:POSetup?	Query power-on setup.		
:VERSion?	Query revision level of SCPI.		✓
:ERRor	Path to read messages in error queue.[1]		✓
[:NEXT]?	Return and clear oldest error (code and message).		
:ALL?	Return and clear all errors (codes and messages).		
:COUNt?	Return the number of errors.		
:CODE	Path to return error code numbers only:		
[:NEXT]?	Return and clear oldest error (code only).		
:ALL?	Return and clear all errors (codes only).		
:CLEar	Clears messages from error queue.		
:KEY <n>	Simulate key-press (1 to 31).		✓
:KEY?	Query the last pressed key.		✓
:AZERo	Control automatic zero:		
[:STATe] <name>	Control automatic zero (ON or OFF).	ON	✓
[:STATe]?	Query state of automatic zero.		✓
:LFRequency <freq>	Select line frequency: 50 or 60 (Hz):		
:LFRequency?	Query line frequency.		
:TIME	Timestamp:		
:RESet	Reset timestamp to zero seconds.		
:LOCal	Take instrument out of remote, cancel remote (RS-232).		
:REMote	Put instrument in remote (RS-232).		
:RWLock	Enable local lockout (RS-232).		

[1]Clearing error queue: Power-up and *CLS clears the error queue. *RST, :SYSTem:PRESet, and :STATus:PRESet have no effect.

Table 16-9
TRACe command summary

Command	Description	Default parameter	SCPI
:TRACe│:DATA	Use :TRACe or :DATA as root command:	Note	✓
:DATA?	Read the contents of the buffer (data store).		✓
:CLEar	Clear readings from buffer.		
:FREE?	Query bytes available and bytes in use.		✓
:POINts <NRf>	Specify size of buffer (1 to 3000).		✓
:ACTual?	Queries number of readings stored in the buffer.		
:POINts?	Query buffer size.		✓
:CONTrol <name>	Specify buffer control mode (NEVER or NEXT).		✓
:CONTrol?	Query buffer control mode.		✓
:TSTamp	Path to set timestamp format:		
:FORMat <name>	Select format (ABSolute or DELTa).		
:FORMat?	Query timestamp format.		

Note: :SYSTem:PRESet and *RST have no effect on the commands in this subsystem.

Table 16-10
TRIGger command summary

Command	Description	Default parameter	SCPI
:INITiate[:IMMediate]	Initiate source and measure cycle(s).		✓
:ABORt	Reset trigger system. Goes to idle state.		✓
:ARM	Path to program Arm Layer:		✓
[:SEQuence[1]]			✓
[:LAYer[1]]			✓
:COUNt <n>	Specify arm count (1 to 3000 or INFinite).	1	✓
:COUNt?	Query arm count (INFinite = +9.9e37).		✓
:SOURce <name>	Specify control source (IMMediate, TIMer, MANual, BUS, TLINk, NSTest, PSTest, BSTest).	IMMediate	✓
:SOURce?	Query control source.		✓
:TIMer <n>	Set timer interval in seconds (0.001 to 99999.99).	0.1	✓
:TIMer?	Query timer interval.		✓
[:TCONfigure]			✓
:DIRection <name>	Enable (SOURce) or disable (ACCeptor) bypass.	ACCeptor	✓
:DIRection?	Query state of bypass.		✓
[:ASYNchronous]	Configure output triggers:		✓
:ILINe <n>	Select input trigger line (1, 2, 3, 4, 5, or 6).	1	
:ILINe?	Query input trigger line.		
:OLINe <n>	Select output trigger line (1, 2, 3, 4, 5, or 6).	2	
:OLINe?	Query output trigger line.		
:OUTPut <name>	Output trigger (TRIGger or NONE).	NONE	
:OUTPut?	Query arm output trigger status.		
:TRIGger	Path to program Trigger Layer:		✓
[:SEQuence[1]]			✓
:COUNt <n>	Specify trigger count (1 to 3000).	1	✓
:COUNt?	Query trigger count.		✓
:DELay <n>	Specify trigger delay: 0 to 999.9999 (sec).	0	✓
:DELay?	Query source delay.		✓
:SOURce <name>	Specify control source (IMMediate or TLINk).	IMMediate	✓
:SOURce?	Query control source.		✓

Table 16-10 (cont.)
TRIGger command summary

Command	Description	Default parameter	SCPI
:TRIGger	Path to program Trigger Layer (continued):		
[:SEQuence[1]]			
[:TCONfigure]			✓
:DIRection <name>	Enable (SOURce) or disable (ACCeptor) bypass.		✓
:DIRection	Query state of bypass.		✓
[:ASYNchronous]	Configure output triggers:		✓
:ILINe <n>	Select input trigger line (1, 2, 3, 4, 5, or 6).	1	
:ILINe?	Query input trigger line.		
:INPut <event list>	Enable input event detectors (SOURce, DELay, SENSe, or NONE).	NONE	
:INPut?	Query enabled input event detectors.		
:OLINe <n>	Select output trigger line (1, 2, 3, 4, 5, or 6).	2	
:OLINe?	Query output trigger line.		
:OUTPut <event list>	Output trigger after SOURce, DELay, SENSe or not (NONE) at all.	NONE	
:OUTPut?	Query when output trigger is going to occur.		

Calculate subsystems

There are a total of eight CALCulate subsystems:

- CALC1: Channel 1 math
- CALC2: Channel 2 math
- CALC3: Channel 1 relative offset
- CALC4: Channel 2 relative offset
- CALC5: Ratio
- CALC6: Delta
- CALC7: Limit tests
- CALC8: Buffer statistics

The commands in these subsystems are summarized in Table 16-1.

NOTE   *See Appendix B for a detailed explanation on how data flows through the various CALC operation blocks of the* Model 6482. *It clarifies the type of readings that are acquired by the various commands to read data.*

CALCulate[1] and CALCulate2

The CALC1 and CALC2 commands control math functions for channel 1 and channel 2 respectively. They perform the remote equivalents to the front panel CONFIG MSR1 and CONFIG MSR2 configuration menus.

Select math function

FORMat <name>

:CALCulate[1]:FORMat <name> Select channel 1 math function

Parameters	<name>	= MXB[1]	MX + B
		COND[1]	I1/V1
		POWER[1]	V1*I1
		RES[1]	V1/I1

Query   :FORMat?   Query selected math function

Description   This command selects the channel 1 math function. The channel 1 current reading (I1) and source voltage (V1) are used to calculate COND1 (conductance), POWER1, and RES1 (resistance), while the MX + B parameter is programmed separately (see below).

FORMat <name>

:CALCulate2:FORMat <name>   Select channel 2 math function

Parameters	<name>	= MXB2	MX + B
		COND2	I2/V2
		POWER2	V2*I2
		RES2	V2/I2

Query   :FORMat?   Query selected math function

Description   This command selects the channel 2 math function. The channel 2 current reading (I2) and source voltage (V2) are used to calculate COND2 (conductance), POWER2, and RES2 (resistance), while the MX + B parameter is programmed separately (see below).

Set MX + B parameters

MBFactor <n>

:CALCulate[1]:KMATh:MBFactor <n>Select channel 1 B (offset) value
:CALCulate2:KMATh:MBFactor <n>Select channel 2 B (offset) value

Parameters	<n>	=	-9.99999e20 to +9.99999e20	Specify B (offset) value for MX + B
			MINimum	-9.99999e20
			MAXimum	+9.99999e20
			DEFault	0

Query   :MBFactor?   Query B (offset) value for MX + B

Description   These commands program the B (offset) value for the MX + B math function (see FORMat <name> above). Use CALC1 for channel 1 or CALC2 for channel 2.

MMFactor <n>

:CALCulate[1]:KMATh:MMFactor <n>Select channel 1 M (slope) value
:CALCulate2:KMATh:MMFactor <n>Select channel 2 M (slope) value

Parameters   <n> =   -9.99999e20 to   Specify M (slope) value for MX + B
                          +9.99999e20
                          MINimum   -9.99999e20
                          MAXimum   +9.99999e20
                          DEFault   1

Query   :MMFactor?   Query M (slope) value for MX + B

Description These commands program the M (slope) value for the MX + B math function (see
                 FORMat <name> above). Use CALC1 for channel 1 or CALC2 for channel 2.

MUNits <name>

:CALCulate[1]:KMATh:MUNits <name>Specify units for channel 1 MX + B
:CALCulate2:KMATh:MUNits <name>Specify units for channel 2 MX + B

Parameters   <name> = Up to three ASCII characters enclosed in single or double quotes

Query   :MUNits?   Query units for MX + B

Description These commands are used to specify the units suffix name for the MX + B math
                 function for channel 1 (CALC1) and channel 2 (CALC2). Use up to three ASCII
                 characters for the units suffix name consisting of upper- or lower-case characters
                 (no numbers, spaces, dashes, etc.).

Enable and read math function result

STATe <b>

:CALCulate[1]:STATe <b>   Control channel 1 math function
:CALCulate2:STATe <b>   Control channel 2 math function

Parameters   <b> =   0 or OFF   Disable CALC1 or CALC2 math
                                           function
                          1 or ON   Enable CALC1 or CALC2 math
                                           function

Query   :STATe?   Query state (on or off) of CALC1 or CALC2

Description These commands are used to enable or disable the CALC1 and CALC2 math
                 function for channels 1 and 2 respectively. When enabled, the selected math
                 function calculation will be performed when the Model 6482 is triggered to perform
                 the programmed source and measure operations.

                 After the Model 6482 returns to idle, you can read the result of the selected math
                 function using the :CALC1:DATA? or :CALC2:DATA? command. (See below).
                 When disabled, the :CALC1:DATA? and CALC2:DATA? command will return the
                 NAN (not a number) value of +9.91e37.

DATA?

:CALCulate[1]:DATA?   Read channel 1 math (CALC1) result
:CALCulate2:DATA?   Read channel 2 math (CALC2) result

Description These query commands are used to read the result of the CALC1 or CALC2
                 calculation. The largest valid calculation result can be ±9.9e37, which (defined by
                 SCPI) is infinity.

An invalid NAN (not a number) result of +9.91e37 indicates that CALC1 or CALC2 is disabled or that a division by zero error has occurred. See :STATe.

LATest?

:CALCulate[1]:DATA:LATest?   Read latest CALC1 result
:CALCulate2:DATA:LATest?   Read latest CALC2 result

Description   These commands work exactly like CALC1:DATA? and CALC2:DATA? except that they return only the latest CALC1 or CALC2 result.

CALCulate3 and CALCulate4

The CALC3 and CALC4 commands configure and control relative for channel 1 and channel 2 respectively. They perform the remote equivalents to the front panel REL and CONFIG REL configuration menu.

Select input path

FEED <name>

:CALCulate3:FEED <name>   Select input path for channel 1 REL

Parameters   <name> = CALCulate[1]   Use result of channel 1 math function
                                    SENSe[1]   Use channel 1 current reading

Query   :FEED?   Query input path for channel 1 REL

Description   This command is used to select the input path for channel 1 REL. With CALCulate[1] selected, the REL feed will be the result of the channel 1 math function (CALC1). With SENSe[1] selected, the REL feed will be the raw channel 1 current reading.

FEED <name>

:CALCulate4:FEED <name>   Select input path for channel 2 REL

Parameters   <name> = CALCulate2   Use result of channel 2 math function
                                    SENSe2   Use channel 2 current reading

Query   :FEED?   Query input path for REL

Description   This command is used to select the input path for channel 2 REL. With CALCulate2 selected, the REL feed will be the result of the channel 2 math function (CALC2). With SENSe2 selected, the REL feed will be the raw channel 2 current reading.

Set or acquire relative value

OFFSet <n>

:CALCulate3:NULL:OFFSet <NRf>Specify null offset (REL) for channel 1
:CALCulate4:NULL:OFFSet <NRf>Specify null offset (REL) for channel 2

Parameters   <NRf> =   -9.999999e20 to Specify null offset value
                          9.999999e20

Query   :OFFSet?   Query null offset value

Description   This command lets you establish a null offset (REL) value for channel 1 (CALC3) or channel 2 (CALC4). When Null Offset is enabled, the result is the algebraic

difference between the feed reading (see
FEED \<name> above) and the offset value:

CALC3 or CALC4 reading = feed reading - null offset

ACQuire

:CALCulate3:NULL:ACQuire	Acquire channel 1 REL value
:CALCulate4:NULL:ACQuire	Acquire channel 2 REL value

Description   These commands automatically acquire the null offset values for channel 1 (CALC3) and channel 2 (CALC4). The next available reading will become the offset value.

Enable and read relative result

STATe \<b>

:CALCulate3:NULL:STATe \<b>	Control channel 1 REL
:CALCulate4:NULL:STATe \<b>	Control channel 2 REL

Parameters   \<b> =   1 or ON   Enable REL
                   0 or OFF   Disable REL

Query   :STATe?   Query state of REL

Description   These commands are used to enable or disable the null or offset value for REL. When enabled, the CALC3 or CALC4 reading will include the null offset value (see OFFSet \<n>). When disabled, the CALC3 or CALC4 reading will not include the null offset.

DATA?

:CALCulate3:DATA?	Read channel 1 REL result
:CALCulate4:DATA?	Read channel 2 REL result

Description   These query commands are used to read the CALC3 or CALC4 REL result. When CALC3 or CALC4 is enabled (see STATe \<b>), the returned value will include the offset value (see OFFSet \<n> and ACQuire).

CALCulate5

The CALC5 commands configure and control the RATIO function. They perform the remote equivalents to the front panel RATIO and CONFIG RATIO configuration menu.

Select RATIO calculation mode

FORMat \<name>

:CALCulate5:FORMat \<name>   Select RATIO calculation mode

Parameters   \<name> = C3C4   CALC3/CALC4
                            C4C3   CALC4/CALC3

Query   :FORMat?   Query selected RATIO calculation

Description   This command selects the RATIO calculation mode. With C3C4 selected, the CALC3 result (channel 1 REL) is divided by the CALC4 result (channel 2 REL). With C4C3 selected, the CALC4 result is divided by the CALC3 result.

Enable and read RATIO result

STATe <b>

	:CALCulate5:STATe <b>	Control RATIO

Parameters   <b> =   1 or ON   Enable RATIO
                        0 or OFF   Disable RATIO

Query   :STATe?   Query state of RATIO

Description   This command is used to enable or disable the RATIO mode. When enabled, the CALC5 reading will be RATIO value determined by FORMat (above). When disabled, or if a divide by zero error occurs, the CALC5 reading will return a NAN (not a number) value of +9.91e37.

DATA?

	:CALCulate5:DATA?	Read RATIO result

Description   This query command is used to read the result of the CALC5 ratio calculation. When enabled, the CALC5 reading will be ratio value determined by FORMat (above). When disabled, or if a divide by zero error occurs, the CALC5 reading will return a NAN (not a number) value of +9.91e37.

CALCulate6

The CALC6 commands configure and control the delta function. They perform the remote equivalents to the front panel DELTA and CONFIG DELTA configuration menu.

Select DELTA calculation mode

FORMat <name>

	:CALCulate6:FORMat <name>	Select DELTA calculation mode

Parameters   <name> = C3C4   CALC3 - CALC4
                           C4C3   CALC4 - CALC3

Query   :FORMat?   Query selected DELTA calculation

Description   This command selects the DELTA calculation mode. With C3C4 selected, the CALC4 result (channel 2 REL) is subtracted from the CALC3 result (channel 1 REL). With C4C3 selected, the CALC3 result is subtracted from the CALC4 result.

Enable and read DELTA result

STATe <b>

	:CALCulate6:STATe <b>	Control DELTA

Parameters   <b> =   1 or ON   Enable DELTA
                        0 or OFF   Disable DELTA

Query   :STATe?   Query state of DELTA

Description   This command is used to enable or disable DELTA mode. When enabled, the CALC6 reading will be DELTA value determined by FORMat (above). When disabled, or if a divide by zero error occurs, the CALC6 reading will return a NAN (not a number) value of +9.91e37.

DATA?

:CALCulate6:DATA?   Read DELTA result

Description   This query command is used to read the result of the CALC5 ratio calculation. When enabled, the CALC5 reading will be ratio value determined by FORMat (above). When disabled, or if a divide by zero error occurs, the CALC5 reading will return a NAN (not a number) value of +9.91e37.

CALCulate7

The CALCulate7 commands configure and control the six limit tests:

- Limit 1: channel 1 compliance (hardware) limit test
- Limit 2: channel 2 compliance (hardware) limit test
- Limit 3 through Limit 6: software limits

Select input path

FEED <name>

:CALCulate7:FEED <name>   Select input path for Limit 3-6 tests

Parameters   <name> = CALCulate3   Use CALC3 result (channel 1 REL)
                            CALCulate4   Use CALC4 result (channel 2 REL)
                            CALCulate5   Use CALC5 result (ratio)
                            CALCulate6   Use CALC6 result (delta)

Query   :FEED?   Query input path for Limit 3-6 tests

Description   This command is used to select the input path for the Limit 3-6 tests (software limits). With CALCulate3 or CALCulate4 selected, the specified limits will be compared to the result of CALC3 (channel 1 REL feed) or CALC4 (channel 2 REL feed). With CALCulate5 selected, the specified limits will be compared with the ratio result. With CALCulate6 selected, the specified limits will be compared with the delta result.

Read limits data

DATA?

:CALCulate7:DATA?   Read limits data

Description   This command is used to acquire all the readings used for the CALC7 limit tests. At least one of the limit tests must be enabled to acquire limit test readings. See "Configure and control limit tests;" "STATe <b>," page Section 16-23.

LATest?

:CALCulate7:DATA:LATest?   Read latest CALC7 data

Description   This command works exactly like CALC7:DATA?, except it returns only the latest limit result.

Configure and control limit tests

COMPliance:FAIL <name>

:CALCulate7:LIMit[1]:COMPliance:FAIL <name>Set channel 1 compliance fail mode
:CALCulate7:LIMit2:COMPliance:FAIL <name>Set channel 2 compliance fail mode

| **Parameters** | <name> = | IN | Fail Limit 1/2 test when instrument goes into compliance |
| | | OUT | Fail Limit 1/2 test when instrument comes out of compliance |

| **Query** | :FAIL? | Query when Limit 1/2 test failure occurs |

Description These commands are used to specify the condition that will cause the Limit 1 and Limit 2 tests to fail. With IN specified, the test will fail when the Model 6482 goes into compliance (fixed 20 mA current limit for each voltage source). With OUT specified, the test will fail when the Model 6482 comes out of compliance.

[:DATA] <n>

:CALCulate7:LIMitx:LOWer[:DATA] <n>Specify lower Limit x (x = 3-6)
:CALCulate7:LIMitx:UPPer[:DATA] <n>Specify upper Limit x (x = 3-6)

Parameters	<n> =	-9.999999e20 to 9.999999e20	Specify limit value
		DEFault	Set specified lower limit to -1
			Set specified upper limit to 1
		MINimum	Set specified limit to -9.999999e20
		MAXimum	Set specified limit to +9.999999e20

Query	:UPPer?	Query specified upper limit
	:UPPer? DEFault	Query *RST default upper limit
	:UPPer? MINimum	Query lowest allowable upper limit
	:UPPer? MAXimum	Query largest allowable upper limit
	:LOWer?	Query specified lower limit
	:LOWer? DEFault	Query *RST default lower limit
	:LOWer? MINimum	Query lowest allowable lower limit
	:LOWer? MAXimum	Query largest allowable lower limit

Description These commands are used to set the upper and lower limits for the Limit 3 through Limit 6 tests (software limits).

STATe <b>

:CALCulate7:LIMit[1]:STATe <b>Control Limit 1 (channel 1) test
:CALCulate7:LIMit2:STATe <b> Control Limit 2 (channel 2) test
:CALCulate7:LIMitx:STATe <b> Control Limit x test (x = 3-6)

| **Parameters** | <b> = | 1 or ON | Enable specified limit test |
| | | 0 or OFF | Disable specified limit test |

| **Query** | :STATe? | Query state of specified limit test |

Description These commands are used to enable or disable Limit 1 through Limit 6, tests. Limit 1 and Limit 2 are the compliance limit tests for channel 1 and channel 2 respectively, while Limit 3 through Limit 6 are the software limit tests. Any limit test not enabled is simply not performed.

FAIL?

:CALCulate7:LIMit[1]:FAIL?	Read Limit 1 (channel 1) test result
:CALCulate7:LIMit2:FAIL?	Read Limit 2 (channel 2) test result
:CALCulate7:LIMitx:FAIL?	Read Limit x test result (x = 3-6)

Description   These commands are used to read the pass/fail results of Limit 1 through Limit 6 tests:

0 = Limit test passed
1 = Limit test failed

The response message (0 or 1) only tells you if a limit test has passed or failed. For Limit 3-6, it does not tell you which limit (upper or lower) has failed. To determine which limit has failed, you must read the measurement event register. See "STATus subsystem," page Section 16-50.

Reading the results of a limit test does not clear the fail indication of the test. A failure can be cleared by using a :CLEar command.

Clear test results

[:IMMediate]

:CALCulate7:CLIMits:CLEar[:IMMediate]Clears test results

Description   This command clears the test results (pass or fail) of the limit tests.

AUTO <b>

:CALCulate7:CLIMits:CLEar:AUTO <b>Control auto-clear for test results

Parameters	<b> =	1 or ON	Enable auto-clear
		0 or OFF	Disable auto-clear

Query   :AUTO?   Query state of auto-clear

Description   With auto-clear enabled, test results will clear when the :INITiate command is sent to start a new test sequence.

When disabled, you must use :IMMediate to perform the clear actions.

CALCulate8

This CALCulate8 commands provide statistical data on buffer readings.

Select statistic

FORMat <name>

:CALCulate8:FORMat <name>   Specify channel 1 buffer format

Parameters	<name> =	MEAN	Mean value of readings in buffer
		SDEViation	Standard deviation of readings in buffer
		MAXimum	Largest reading in buffer
		MINimum	Lowest reading in buffer
		PKPK	MAXimum - MINimum

Query   :FORMat?   Query programmed buffer math format

Description   This command is used to select the statistic on readings stored in the channel 1 buffer. See Section 6 for details on these statistics.

NOTE   *Buffer statistics are performed on the presently selected display function (MSR1, MSR2, RATIO, or DELTA) selected with the :DISPlay:MODE command. Buffer statistics cannot be performed in the dual-channel display mode.*

Acquire statistic

DATA?

:CALCulate8:DATA?   Read channel 1 buffer statistic result

Description   This query command is used to perform the selected statistic operation and read the result(s). The result(s) is always returned in ASCII format.

Statistic operations are not performed on TIME and STATus data elements that are stored in the buffer.

Notes:

1. If there is no data in the buffer, error -230, "Data corrupt or stale," will be generated.

2. If there are a lot of readings stored in the buffer, some statistic operations may take too long and cause a bus time-out error. To avoid this situation, send the :calc8:data? command and then wait for the MAV (message available) bit in the status byte register to set before addressing the Model 6482 to talk. See Section 13.

3. See Appendix B for a detailed explanation on how data flows through the various operation blocks of the Model 6482. It clarifies the types of readings that are acquired by the various commands to read data.

DISPlay subsystem

The display subsystem controls the display of the Model 6482 and is summarized in Table 16-2.

Control display

DIGits <n>

:DISPlay:DIGits <n>   Set display resolution

Parameters   <n> =

	4	3.5 digit resolution
	5	4.5 digit resolution
	6	5.5 digit resolution
	7	6.5 digit resolution
	DEFault	5.5 digit resolution
	MINimum	3.5 digit resolution
	MAXimum	6.5 digit resolution

Query

:DIGits?   Query display resolution
:DIGits? DEFault   Query *RST default resolution
:DIGits? MINimum   Query lowest allowable display resolution
:DIGits? MAXimum   Query largest allowable display resolution

Description   This command is used to set the display resolution. Note that you can instead use rational numbers. For example, to select 4.5 digit resolution, you can send a parameter value of 4.5 (instead of 5). The Model 6482 rounds the rational number to an integer.

ENABle <b>

	:DISPlay:ENABle <b>	Control display circuitry

Parameters   <b> =   0 or OFF   Disable display circuitry
    1 or ON   Enable display circuitry

Query   :ENABle?   Query state of display

Description   This command is used to enable and disable the front panel display circuitry. When disabled, the instrument operates at a higher speed. While disabled, the display is frozen with the following message:

FRONT PANEL DISABLED
Press LOCAL to resume.

As reported by the message, all front panel controls (except LOCAL and OUTPUT OFF) are disabled. Normal display operation can be resumed by using the :ENABle command to enable the display or by putting the Model 6482 into local.

MODE <name>

	:DISPlay:MODE <name>	Select CALC block for display

Parameters   <name> =   CALCulate3   Display CALC3 (Channel 1 REL)
    CALCulate4   Display CALC4 (Channel 2 REL)
    CALCulate5   Display CALC5 (RATIO)
    CALCulate6   Display CALC6 (DELTA)
    DUAL   Dual-channel display mode

Query   :MODE?   Query selected display CALC block

Description   This command is used to select which CALC block to display. Displayed CALC blocks include CALC3 (channel 1 REL), CALC4 (channel 2 REL), CALC5 (RATIO), and CALC6 (DELTA). The DUAL parameter selects the dual-channel display mode.

ATTRibutes?

	:DISPlay[:WINDow[1]]:ATTRibutes?	Query attributes; top display
	:DISPlay:WINDow2:ATTRibutes?	Query attributes; bottom display

Description   This query command is used to determine which characters on the display are blinking and which are not. The response message provides that status of each character position for the specified display. The primary display consists of 20 characters and the secondary display consists of 32 characters.

1 = Character is blinking
0 = Character is not blinking

For example, assume the following menu is displayed with the SAVESETUP option blinking:

MAIN MENU
SAVESETUP COMMUNICATION CAL >

The response message for :disp:attr? (top display) will result in 20 zeroes as follows:

00000000000000000000

The response message for :disp:wind2:attr? (bottom display) will display ones at the character positions for SAVESETUP as follows:

11111111100000000000000000000000

Read display

DATA?

:DISPlay[:WINDow[1]]:DATA?   Read top display
:DISPlay:WINDow2:DATA?   Read bottom display

Description   These query commands are used to read what is currently being displayed on the top and bottom displays. After sending one of these commands and addressing the Model 6482 to talk, the displayed data (message or reading) will be sent to the computer.

Define :TEXT messages

DATA <a>

:DISPlay[:WINDow[1]]:TEXT:DATA <a>Define message; top display
:DISPlay:WINDow2:TEXT:DATA <a>Define message; bottom display

Parameters   <a> = ASCII characters for message
  Types: String   'aa...a' or "aa...a"
             Indefinite Block   #0aa...a
             Definite Block   #XYaa...a
  where   Y = number of characters in message:
             Up to 20 for top display
             Up to 32 for bottom display
             X = number of digits that make up Y (1 or 2)

Query   :DATA?   Query the defined text message

Description   These commands define text messages for the display. A message can be as long as 20 characters for the top display, and up to 32 characters for the bottom display. A space is counted as a character. Excess message characters result in an error.

An indefinite block message must be the only command in the program message or the last command in the program message. If you include a command after an indefinite block message (on the same line), it will be treated as part of the message and is displayed instead of executed.

STATe <b>

:DISPlay[:WINDow[1]]:TEXT:STATe <b>Control message; top display
:DISPlay:WINDow2:TEXT:STATe <b>Control message; bottom display

Parameters   <b> =   0 or OFF   Disable message for specified display
                                1 or ON   Enable message for specified display

Query   :STATe?   Query state of message mode for specified display

Description   These commands enable and disable the text message modes. When enabled, a defined message is displayed. When disabled, the message is removed from the display.

GPIB operation: A user defined text message remains displayed only if the instrument is in remote. Taking the instrument out of remote (by pressing the LOCAL key or sending LOCAL 25) cancels the message and disables the text message mode.

RS-232 operation: A user defined test message can be cancelled by sending the :SYSTem:LOCal command or pressing the LOCAL key.

FORMat subsystem

The commands for this subsystem are used to select the data format for transferring instrument readings over the bus. These commands are summarized in Table 16-3.

Data format

[:DATA] <type>[,length]

:FORMat[:DATA] <type>[,<length>]Select data format

Parameters   <type>[,<length>] =ASCiiASCII format
                    REAL,32   IEEE754 single precision format
                    SREal   IEEE754 single precision format

NOTE   *<length> is not used for the ASCii or SREal parameters. It is optional for the REAL parameter. If you do not use <length> with the REAL parameter, the <length> defaults to 32 (single precision format).*

Query   [:DATA]?   Query data format

Description   This command is used to select the data format for transferring readings over the bus. Only the ASCII format is allowed over the RS-232 interface. This command only affects the output of READ?, FETCh?, MEASure?, TRACe:DATA?, CALCx:DATA? over the GPIB. All other queries are returned in the ASCII format.

NOTE   *Regardless of which data format for output strings is selected, the* Model 6482 *will only respond to input commands using the ASCII format.*

ASCII format

The ASCII data format is in a direct readable form for the operator. Most BASIC languages easily convert ASCII mantissa and exponent to other formats. However, some speed is compromised to accommodate the conversion. Figure 16-1 shows an example ASCII string that includes current and voltage for one channel, as well as time, and status data elements. See ELEMents <item list>.

Figure 16-1 also shows the byte order of the data string. Data elements not specified by the :ELEMents command are simply not included in the string.

Keep in mind that the byte order can only be reversed for the binary format. See BORDer <name>.

Figure 16-1
ASCII data format

```
+1.000206E-03,    +1.000000E-02,  +7.282600E+01,  +4.813200E+04
```

Channel 1	Channel 2	Time	Status
Reading	Reading		
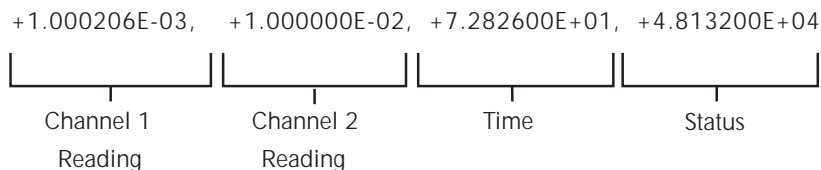
IEEE-754 single precision format

REAL, 32, or SREal will select the binary IEEE-754 single precision data format. Figure 16-2 shows the normal byte order format for each data element. For example, if three valid elements are specified, the data string for each reading conversion is made up of three 4-byte data blocks. Note that the data string for each reading conversion is preceded by a 2-byte header that is the binary equivalent of an ASCII # sign and 0. Figure 16-2 does not show the byte for the terminator that is attached to the end of each data string.

Figure 16-2
IEEE-754 single precision data format (32 data bits)
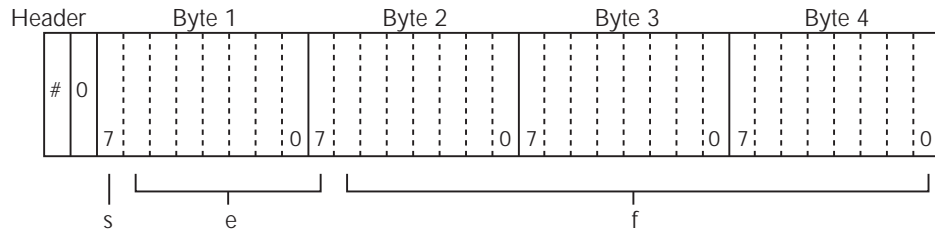
s = sign bit (0 = positive, 1 = negative)
e = exponent bits (8)
f = fraction bits (23)

Normal byte order shown. For swapped byte order, bytes sent
in reverse order: Header, Byte 4, Byte 3, Byte 2, Byte 1.

The header and terminator are sent only once for each READ?

During binary transfers, never un-talk the Model 6482 until after the data is read (input) to the computer. Also, to avoid erratic operation, the readings of the data string (and terminator) should be acquired in one piece. The header (#0) can be read separately before the rest of the string.

NOTE   *The Model 6482 terminates the binary data string with LF + EOI. If your program is set to terminate on CR and/or LF, data transfer may terminate prematurely because one or more data bytes has a CR or LF value. To avoid the problem, set your computer program to terminate on EOI only.*

The number of bytes to be transferred can be calculated as follows:

Bytes = $2 + (\text{Rdgs} \times 4) + 1$

where 2 is the number of bytes for the header (#0).
  Rdgs is the product of the number of selected data elements, arm
    count, and trigger count.
  4 is the number of bytes for each reading.
  1 is the byte for the terminator.

For example, assume the Model 6482 is configured to perform 10 source and measure operations and send the 10 current measurements to the computer using the binary format.

Bytes = 2 + (10 x 4) + 1
     = 43

Data elements

ELEMents <item list>

:FORMat:ELEMents [SENSe] <item list>Specify data elements for data string

Parameters   <item list> =CURRent[1]   Includes channel 1 current reading
                    CURRent2   Includes channel 2 current reading
                    TIME   Includes timestamp
                    STATus   Includes status information

NOTE   *Each item in the list must be separated by a comma (for example, CURR1, CURR2, TIME).*

Query   :ELEMents?   Query elements in data string

Description   This command is used to specify the elements to be included in the data string in response to the following queries:

:FETCh?
:READ?
:MEASure?
:TRACe:DATA?

You can specify from one to all four elements. Each element in the list must be separated by a comma (,). These elements (shown in Figure 16-1) are explained as follows:

NOTE   *An overflow reading reads as +9.9E37.*

CURRent[1]: This element provides the channel 1 current reading. If no reading is available, the NAN (not a number) value of +9.91e37 is used.

CURRent2: This element provides the channel 2 current reading. If no current reading is available, the NAN (not a number) value of +9.91e37 is used.

TIME: A timestamp is available to reference each group of readings to a point in time. The relative timestamp operates as a timer that starts at zero seconds when the instrument is turned on or when the relative timestamp is reset (:SYSTem:TIME:RESet). The timestamp for each reading sent over the bus is referenced, in seconds, to the start time. After 99,999.999 seconds, the timer resets to zero and starts over.

Timestamp is also available for buffer readings. Timestamp can be referenced to the first reading stored in the buffer (absolute format) which is timestamped at 0 seconds, or can provide the time between each reading (delta format). The :TRACe:TSTamp:FORMat command is used to select the absolute or delta format.

NOTE   *Timestamp values are approximate. See Section 6 for details.*

STATus: A status word is available to provide status information concerning Model 6482 operation. The 24-bit status word is sent in a decimal form and must be converted by the user to the binary equivalent to determine the state of each bit in the word. For example, if the status value is 65, the binary equivalent is 0000000000001000001. Bits 0 and 6 are set.

The significance of each status bit is explained as follows:

Bit 0 (OFLO1): Set to 1 if channel 1 measurement was made while in over-range. Cleared to 0 otherwise.

Bit 1 (OFLO2): Set to 1 if channel 2 measurement was made while in over-range. Cleared to 0 otherwise.

Bit 2 (FILTERED): Set to 1 if either channel 1 or channel 2 measurement was made with the filter enabled. Cleared to 0 otherwise

Bit 3 (COMPLIANCE1): Set to 1 if the channel 1 voltage source is in compliance. Cleared to 0 otherwise.

Bit 4 (COMPLIANCE2): Set to 1 if the channel 2 voltage source is in compliance. Cleared to 0 otherwise.

Bit 5 (NULL1_ACTIVE): Set to 1 if channel 1 REL (CALC3) is enabled. Cleared to 0 otherwise.

Bit 6 (NULL2_ACTIVE): Set to 1 if channel 2 REL (CALC4) is enabled. Cleared to 0 otherwise.

Bit 7 (LIMITS_ACTIVE): Set to 1 if a limit test (CALC7) is enabled. Cleared to 0 otherwise.

Bit 8 (LIMIT_A): Encoded limit test summary bit. See grading and sorting modes in "Limit test bits" below.

Bit 9 (LIMIT_B): Encoded limit test summary bit. See below.

Bit 10 (LIMIT_C): Encoded limit test summary bit. See below.

Bit 11 (LIMIT_D): Encoded limit test summary bit. See below.

Bit 12 (LIMIT_HIGH): Limit test 3 - 6 failed high. See below.

Bit 13 (CH1_OUTPUT): Set to 1 if the channel 1 source output is on. Cleared to 0 otherwise.

Bit 14 (CH2_OUTPUT): Set to 1 if the channel 2 source output is on. Cleared to 0 otherwise.

Bit 15: Not used.

Limit test bits   Bits 8 through 12 flag pass/fail conditions for the various limit tests. The bit values for the grading and sorting modes are covered below. See :CALC7:CLIM:MODE and associated commands in "Calculate subsystems," page Section 16-16.

Grading mode status bit values:

Result Bit #:	12	11	10	9	8	Meas. Event Status[1]
All limit tests passed	0	0	0	0	0	Bit 5 (LP)
Limit test 1 failed	0	0	0	0	1	Bit 0 (L1)
Limit test 2 failed	0	0	0	1	0	Bit 1 (L2)
Limit test 3 high failed	1	0	0	1	1	Bit 4 (LFH)
Limit test 3 low failed	0	0	0	1	1	Bit 2 (L3)
Limit test 4 high failed	1	0	1	0	0	Bit 4 (LFH)
Limit test 4 low failed	0	0	1	0	0	Bit 3 (L4)
Limit test 5 high failed	1	0	1	0	1	-
Limit test 5 low failed	0	0	1	0	1	-
Limit test 6 high failed	1	0	1	1	0	Bit 4 (LFH)
Limit test 6 low failed	0	0	1	1	0	-

[1]See Section 13, "Measurement event register," and Figure 13-6 for details.

Sorting mode status bit values:

Result Bit #:	12	11	10	9	8	Meas. Event Status[1]
Limit 1 & 2 passed, 3-6 disabled	0	0	0	0	0	Bit 5 (LP)
Limit test 1 failed	0	0	0	0	1	Bit 0 (L1)
Limit test 2 failed	0	0	0	1	0	Bit 1 (L2)
Limit test 3 passed	0	0	0	1	1	Bit 3 (L3)
Limit test 4 passed	0	0	1	0	0	Bit 4 (L4)
Limit test 5 passed	0	0	1	0	1	-
Limit test 6 passed	0	0	1	1	0	-
All Limits 3 - 6 failed, Limits 1 and 2 passed	1	1	1	1	1	-

[1]See Section 13, "Measurement event register," and Figure 13-6 for details.

SOURce3 <name>

:FORMat:SOURce3 <name>   Set SOUR3 and TTL response formats

Parameters   <name> = ASCii   ASCII format
                             HEXadecimal   Hexadecimal format
                             OCTal   Octal format
                             BINary   Binary format

Query   :SOURce3?   Query response format

Description   This command controls the response format for all CALC7:XXXX:SOUR3 and SOUR3:TTL queries in a manner similar to formats set by the FORM:SREG command. See "Calculate subsystems," and "SOURce subsystem," page Section 16-40, topics for details.

CALC data elements

CALCulate <item list>

:FORMat:ELEMents:CALCulate <item list>Set CALC data elements

Parameters   <item list> =CALCulate   Include CALC1 to CALC7 data
                               TIME   Include timestamp
                               STATus   Include status information

NOTE   *Each item in the list must be separated by a comma (for example, :CALCulate CALC,TIME,STAT).*

Query   :CALCulate?   Query CALC data element list

Description   This command allows you specify the data elements returned with the CALC1:DATA? through CALC7:DATA? queries. With CALC specified, the corresponding CALC1 through CALC7 element will be included in the response. With TIMe or STATus specified, timestamp and status information will be returned with the queries. See "Calculate subsystems," page Section 16-16, for a complete description of the various CALC1 through CALC7 elements. See also "Data elements," page Section 16-29, for a description of TIME and STATus information.

TRACe data elements

TRACe <item list>

:FORMat:ELEMents:TRAce <item list>Specify data elements for buffer

Parameters   <item list> =CURRent[1]   Includes channel 1 current reading
                               CURRent2   Includes channel 2 current reading
                               CALCulate1   Includes CALC1 element
                               CALCulate2   Includes CALC2 element
                               CALCulate3   Includes CALC3 element
                               CALCulate4   Includes CALC4 element
                               CALCulate5   Includes CALC5 element
                               CALCulate6   Includes CALC6 element
                               CALCulate7   Includes CALC7 element
                               TIME   Includes timestamp
                               STATus   Includes status information
                               ALL   Includes all data elements
                               DEFault   Includes only CURR1 and CURR2

NOTE   *Each item in the list must be separated by a comma (for example, CURR1, CALC2, TIME).*

Query	:TRACe?	Query elements in data string
Description	This command is used to specify the elements to be recalled from the data store buffer (see "TRACe subsystem," page Section 16-56). You can specify from one to all 11 elements, or use ALL to specify all elements. Each element in the list must be separated by a comma (,). These elements are explained as follows:	

NOTE   *An overflow reading reads as +9.9E37.*

CURRent[1]: This element provides the channel 1 current reading. If no reading is available, the NAN (not a number) value of +9.91e37 is used.

CURRent2: This element provides the channel 2 current reading. If no current reading is available, the NAN (not a number) value of +9.91e37 is used.

CALCulate1 ... CALCulate7: These elements include the result of the CALC1 through CALC7 calculations. If no value is available, the NAN (not a number) value of +9.91e37 is used.

TIME: This element includes the timestamp value. See "Data elements," page Section 16-29, for complete description.

STATus: This element includes the 24-bit status word. See "Data elements," page Section 16-29, for a complete description.

ALL: This parameter includes all CURR, CALC, TIME, and STAT elements.

DEFault: This parameter includes only the CURRent[1] and CURRent2 elements.

Byte order

BORDer <name>

	:FORMat:BORDer <name>	Specify binary byte order
Parameters	<name> = NORMal	Normal byte order for binary formats
	SWAPped	Reverse byte order for binary formats
Query	:BORDer?	Query byte order
Description	This command is used to control the byte order for the IEEE-754 binary formats. For normal byte order, the data format for each element is sent as follows:	

Byte 1   Byte 2   Byte 3   Byte 4   (Single precision)

For reverse byte order, the data format for each element is sent as follows:

Byte 4   Byte 3   Byte 2   Byte 1   (Single precision)

The "#0" header is not affected by this command. The header is always sent at the beginning of the data string for each measurement conversion.

The ASCII data format can only be sent in the normal byte order. The SWAPped selection is simply ignored when the ASCII format is selected.

NOTE   *The SWAPped byte order must be used when transmitting binary data to any IBM PC compatible computer.*

Status register format

SREGister <name>

:FORMat:SREGister <name>   Set data format for reading status registers

Parameters   <name> = ASCii   Decimal format
                            Hexadecimal   Hexadecimal format
                            OCTal   Octal format
                            BINary   Binary format

Query   :SREGister?   Query format for reading status registers

Description   Query commands are used to read the contents of the status event registers. This command is used to set the response message format for those query commands.

When a status register is queried, the response message is a value that indicates which bits in the register are set. For example, if bits B5, B4, B2, B1, and B0 of a register are set (110111), the following values will be returned for the selected data format:

ASCii   55   (decimal value)
Hexadecimal   #H37   (hexadecimal value)
OCTal   #Q67   (octal value)
BINary   #B110111 (binary value)

See Section 13 and "Status Structure" in this section for more information.

OUTPut subsystem

This subsystem is used to control the voltage source outputs and the output enable line. These commands are summarized in Table 16-4.

Turn source on or off

[:STATe] <b>

:OUTPut[1][:STATe] <b>   Turn source 1 on or off
:OUTPut2[:STATe] <b>   Turn source 2 on or off

Parameters   <b> =   0 or OFF   Turn source off (standby)
                      1 or ON   Turn source on (operate)

Query   :OUTPut[1]?   Query state of source 1
                :OUTPut2?   Query state of source 2

Description   These commands are used to turn the channel 1 and channel 1 source outputs on or off.

Turning a source off places the Model 6482 in the idle state. The only exception to this is when source auto clear is enabled. In this mode, the source turns on during each source phase of the SDM cycle and then turns off after each measurement. See :SOURce[1]:CLEar:AUTO and :SOURce2:CLEar:AUTO in "SOURce subsystem," page Section 16-40.

NOTE   *The :SOURce[1]:CLEar or :SOURce2:CLEar command will also turn the respective source off.*

Output enable control

ENABle[:STATe] <b>

:OUTPut[1]:ENABle[:STATe] <b>Control output enable for both sources

Parameters <b> = 0 or OFF   Disable output enable
                         1 or ON   Enable output enable

Query   :ENABle[:STATe]?   Query state of output enable

Description This command is used to enable or disable the output enable line. When enabled, the sources cannot be turned on unless the output enable line (pin 8 of the rear panel OUTPUT ENABLE connector) is pulled to a logic low state. When the output enable line goes to a logic high state, both sources turn off. See Section 11, "Output enable" and "Output enable," for details using output enable with a test fixture.

When disabled, the logic level on the output enable line has no effect on the output state of the source.

TRIPped?

:OUTPut[1]:ENABle:TRIPped?

Description This query command is used to determine if output enable has been tripped. The tripped condition ("1") means that the sources can be turned on (output enable line at logic low level).

A "0" will be returned if the sources cannot be turned on (output enable line at logic high level).

SENSe subsystem

The SENSe subsystem is used to configure and control the measurement functions of the Model 6482. SENSe[1] controls channel 1 measurement functions, and :SENSe2 controls channel 2 functions. Some of the commands are global, where a single command affects both channels. Other commands are unique to a specific channel. For example, you can program a unique range setting for each channel. A channel does not have to be selected before you can program its various configurations. Whenever a programmed channel is selected, it assumes the programmed states.

The commands for this subsystem are summarized in Table 16-5.

Select measurement range

[:UPPer] <n>

[:SENSe[1]]:CURRent[:DC]:RANGe[:UPPer] <n>|UP|DOWNSelect range for channel 1
:SENSe2:CURRent[:DC]:RANGe[:UPPer] <n>|UP|DOWNSelect range for channel 2

Parameters <n> = -21e-3 to 21e-3 Expected reading in amps
                         DEFault   2e-4
                         MINimum   -21e-3
                         MAXimum   21e-3
                         UP   Select next higher measurement range
                         DOWN   Select next lower measurement range

Query	:RANGe?	Query measurement range
	:RANGe? DEFault	Query *RST default range
	:RANGe? MINimum	Query lowest range (returns 0)
	:RANGe? MAXimum	Query highest range

Description   These commands are used to manually select the measurement range for channel 1 and channel 2. The range is selected by specifying the expected reading. The instrument will then go to the most sensitive reading that will accommodate that reading. For example, if you expect a reading of approximately 5 mA, then simply let <n> = 0.005 (or 5e-3) in order to select the 20 mA range.

You can also use the UP and DOWN parameters to select range. Each time UP or DOWN is sent, the next higher or lower measurement range is selected. When on the maximum range, sending UP is a NO-Op (no operation). When on the lowest range, sending DOWN is a NO-Op.

Measurement ranges can instead be automatically selected by the instrument. See "AUTO <b>," page Section 16-36.

Select autorange

AUTO <b>

[:SENSe[1]]:CURRent[:DC]:RANGe:AUTO <b>Control autoranging for channel 1
:SENSe2:CURRent[:DC]:RANGe:AUTO <b>Control autoranging for channel 2

Parameters	<b> =	0 or OFF	Disable autorange
		1 or ON	Enable autorange

Query	:AUTO?	Query state of autorange

Description   This command is used to control autoranging. With autoranging enabled, the instrument automatically goes to the most sensitive range to perform the measurement.

When this command is used to disable autorange, the instrument remains at the automatically selected range. When a range is manually selected, autorange is disabled. See the previous command.

LLIMit <n>

[:SENSe[1]]:CURRent[:DC]:RANGe:AUTO:LLIMit <n>Set autoranging lower limit for channel 1
:SENSe2:CURRent[:DC]:RANGe:AUTO:LLIMit <n>Set autoranging lower limit for channel 2

Parameters	<n> =	-21e-3 to 21e-3	Lower limit
		DEFault	2e-9
		MINimum	0
		MAXimum	20e-3

Query	:LLIMit?	Query autorange lower limit

Description   The lower limit programmed by these commands for both channels must be less than or equal to the upper limit. If the lower limit is equal to the upper limit, autoranging is effectively disabled. See below. When autoranging is disabled, you can manually program the instrument for any range above the lower limit.

ULIMit <n>

[:SENSe[1]]:CURRent[:DC]:RANGe:AUTO:ULIMit <n>Set autoranging upper limit for channel 1
:SENSe2:CURRent[:DC]:RANGe:AUTO:ULIMit <n>Set autoranging upper limit for channel 2

Parameters	<n> =	-21e-3 to 21e-3	Upper limit
		DEFault	20e-3
		MINimum	0
		MAXimum	20e-3

Query   :ULIMit?   Query autorange upper limit

Description The upper limit programmed by these commands for both channels must be greater than or equal to the lower limit. If the upper limit is equal to the lower limit, autorange is disabled. When autoranging is disabled, you can manually program the instrument for any range below the upper limit.

Set measurement speed

NPLCycles <n>

[:SENSe[1]]:CURRent[:DC]:NPLCycles <n>Set channel 1 speed (PLC)
:SENSe2:CURRent[:DC]:NPLCycles <n>Set channel 2 speed (PLC)

Parameters	<n> =	0.01 to 10	Power-line cycles per integration
		DEFault	1
		MINimum	0.01
		MAXimum	10

Query   :NPLCycles?   Query programmed PLC value
                 :NPLCycles? DEFault   Query *RST default PLC
                 :NPLCycles? MINimum Query minimum PLC
                 :NPLCycles? MAXimumQuery maximum PLC

Description This command is used to set the integration period (speed) for measurements. NPLC (Number of Power Line Cycles) expresses the integration period by basing it on the power line frequency. For example, for a PLC of 1, the integration period would be 1/60 (for 60Hz line power) which is 16.67 msec.

Note that this is a global command. Thus, if you set the speed for channel 1 measurements to 10 PLC, then channel 2 will also set to 10 PLC.

Configure and control filters

NOTE   *Detailed information on the average and median filters are provided in Section 4, "Filters."*

All filter commands are global and commands affect both channels simultaneously.

Average filter commands

COUNt <n>

[:SENSe[1]]:AVERage:COUNt <n>Set average filter count
:SENSe2:AVERage:COUNt <n> Set average filter count

Parameters	<n> =	1 to 100	Specify average filter count
		DEFault	10
		MINimum	1
		MAXimum	100

Query	:COUNt?	Query filter count
	:COUNt? DEFault	Query the *RST default filter count
	:COUNt? MINimum	Query the lowest allowable filter count
	:COUNt? MAXimum	Query the largest allowable filter count

Description   These commands are used to specify the average filter count for channel 1 and channel 2. In general, the filter count is the number of readings that are acquired and stored in the filter buffer for the averaging calculation. Each aquired group of readings yields a single filtered reading. The larger the filter count, the more filtering that is performed.

[:STATe] <b>

[:SENSe[1]]:AVERage:[:STATe] <b>Enable or disable average filter
:SENSe2:AVERage:[:STATe] <b>Enable or disable average filter

Parameters	<b> =	0 or OFF	Disable average filter
		1 or ON	Enable average filter

Query	[:STATe]?	Query state of average filter

Description   These commands are used to enable or disable the average filter. When enabled, current readings are filtered according to how the average filter is configured (see TCONtrol <name>). When disabled, the average filter stage is bypassed.

TCONtrol <name>

[:SENSe[1]]:AVERage:TCONtrol <name>Set average filter type
:SENSe2:AVERage:TCONtrol <name>Set average filter type

Parameters	<name> =	MOVing	Moving average filter
		REPeat	Repeat average filter

Query	:TCONtrol?	Query type of average filter

Description   These commands are used to control the type of average filter. The REPeat filter places the specified number of measurement conversions into a stack and averages them to yield a single repeat filter reading. The stack is then cleared, and the process starts over.
The MOVing average filter uses a first-in, first-out stack. When the stack (filter count) becomes full, the readings are averaged, yielding a filtered reading. For each subsequent reading placed into the stack, the oldest reading is discarded. The stack is reaveraged, yielding a new reading.

ADVanced:NTOLerance <n>

[:SENSe[1]]:AVERage:ADVanced:NTOLerance <n>Set filter noise window
:SENSe2:AVERage:ADVanced:NTOLerance <n>Set filter noise window

Parameters	<n> =	0 to 105	Specify filter noise tolerance in %
		DEfault	5
		MINimum	0
		MAXimum	100

Query   :NTOLerance?   Query filter noise tolerance value

Description   When the advanced filter is enabled (see "STATe" below) a noise window is used with the average filter. This command is used to specify the noise window. If readings are within the noise window, the average filter operates normally. If, however, a reading falls outside the window, the buffer is flushed of old readings and filled with the new reading.

ADVanced[:STATe] <b>

[:SENSe[1]]:AVERage:ADVanced[:STATe] <b>Enable or disable advanced filter
:SENSe2:AVERage:ADVanced[:STATe] <b>Enable or disable advanced filter

Parameters	<b> =	0 or OFF	Disable advanced filter
		1 or ON	Enable advanced filter

Query   [:STATe]?   Query state of advanced filter

Description   This command is used to enable or disable the advanced filter. When enabled, the noise window is used with the averaging filter. When disabled, the noise window is not used.

Median filter commands

MEDian:RANK <NRf>

[:SENSe[1]]:MEDian:RANK <n> Set median filter rank
:SENSe2:MEDian:RANK <n>   Set median filter rank

Parameters	<n> =	0 to 5	Specify rank value for median filter
		DEFault	1
		MINimum	0
		MAXimum	5

Query   RANK?   Query median filter rank

Description   The median filter is used to pass the middle-most reading from a group of readings that are arranged according to size. This command is used to specify the rank, which determines the number of reading samples for the filter process.

The number of reading samples are determined as follows:

Sample readings = 2n + 1

Where: n is the selected rank (1 to 5).

MEDian[:STATe] <b>

[:SENSe[1]]:MEDian[:STATe] <b>Enable or disable channel 1 median filter
:SENSe2:MEDian[:STATe] <b>   Enable or disable channel 2 median filter

Parameters	<b> =	0 or OFF	Disable repeat filter
		1 or ON	Enable repeat filter

Query   [:STATe]?   Query state of repeat filter

Description   This command is used to enable or disable the median filter. When enabled, current readings are filtered according to the specified rank. When disabled, the median filter stage is bypassed.

SOURce subsystem

This subsystem is used to configure and control the two voltage sources. The commands for this subsystem are summarized in Table 16-6.

SOURce[1] and SOURce2

Use the following commands to configure and control the channel 1 and channel 2 voltage sources. SOURce[1] controls the channel 1 source, and SOURce2 controls the channel 2 source.

Control source output on-off

NOTE   *All SOURx:CLEar commands are global and affect both channels simultaneously.*

[:IMMediate]

:SOURce[1]:CLEar[:IMMediate] Turn source output off

Description   This command is used to turn off the channel 1 and channel 2 source outputs. The outputs will turn off after all programmed source and measure operations are completed and the instrument returns to the idle state.

Note that if auto output-off is enabled, the source outputs will automatically turn off. See the next command.

MODE <name>

:SOURce[1]:CLEar:AUTO:MODE <name>Control source auto on/off mode
:SOURce2:CLEar:AUTO:MODE <name>Control source auto on/off mode

Parameters   <name> = ALWays   Output on before reading, off after
                                               reading
                                  TCOunt   Output on trigger layer entry, off on
                                               trigger layer exit

Query   :MODE?   Query state of auto on/off

Description   These commands are used to control the source auto on/off output mode for the two sources. With ALWays selected, the source output will turn on before each reading and then off after each reading. With TCOunt selected, the source will turn on when entering the trigger layer and off when exiting the trigger layer. See Section 8 for details on the trigger model.

Select sourcing mode

MODE <name>

:SOURce[1]:VOLTage:MODE <name>Select sourcing mode for source 1
:SOURce2:VOLTage:MODE <name>Select sourcing mode for source 2

Parameters   <name> = FIXed   Select fixed sourcing mode
                                  LIST   Select list sourcing mode
                                  SWEep   Select sweep sourcing mode

Query	:MODE?		Query DC sourcing mode

Description   These commands are used to select the sourcing mode for the specified source. The three modes are explained as follows:

FIXed: In this sourcing mode, the specified source will output a fixed level. Use the :RANGe and :AMPLitude commands to specify the fixed source level. See "Select range," page Section 16-41, and "Set amplitude for fixed source," page Section 16-42.

LIST: In this mode, the source will output levels that are specified in a list. See "Configure list," page Section 16-48, for commands to define and control the execution of the list.

SWEep: In this mode, the source will perform a voltage sweep. See "Configure sweeps," page Section 16-44, for commands to define the sweep.

NOTE   *The sourcing mode will default to FIXed whenever the* Model 6482 *goes to the local state.*

Select range

RANGe <n>

:SOURce[1]:VOLTage:RANGe <n>Select range for channel 1 source
:SOURce2:VOLTage:RANGe <n>Select range for channel 2 source

Parameters	<n> =	-30 to 30	Specify source level (volts)
		DEFault	10 V range
		MINimum	10 V range
		MAXimum	30 V range
		UP	Select next higher range
		DOWN	Select next lower range

Query	:RANGe?	Query range for specified source
	:RANGe? DEFault	Query *RST default source range
	:RANGe? MINimum	Query lowest source range
	:RANGe? MAXimum	Query highest source range

Description   These commands are used to manually select the range for the specified source. Range is selected by specifying the approximate source magnitude that you will be using. The instrument will then go to the lowest range that can accommodate that level. For example, if you expect to source levels around 3 V, send the following command:

:SOURce1:VOLTage:RANGe 3

The above command will select the 10 V range for the channel 1 source.

As listed in "Parameters," page 17-61, you can also use the MINimum, MAXimum and DEFault parameters to manually select the source range. The UP parameter selects the next higher source range, while DOWN selects the next lower source range.

Note that source range can be selected automatically by the instrument. See the next command.

AUTO <b>

:SOURce[1]:VOLTage:RANGe:AUTO <b>Select autorange for source 1
:SOURce2:VOLTage:RANGe:AUTO <b>Select autorange for source 2

Parameters	<b> =	0 or OFF	Disable autorange
		1 or ON	Enable autorange

Query   AUTO?   Query state of autorange

Description   These commands are used to enable or disable autorange for the specified source. When enabled, the instrument will automatically select the most sensitive range for the specified source level. When disabled, the instrument will use the range that the instrument is currently on.

Autorange will be disabled if a fixed range is selected. See the previous command.

Both *RST and :SYSTem:PREset enables source autorange. When the Model 6482 goes into the local state, source autorange disables.

Set amplitude for fixed source

[:IMMediate][:AMPLitude] <n>

:SOURce[1]:VOLTage[:LEVel][:AMPLitude] <n>Set fixed source 1 amplitude
:SOURce2:VOLTage[:LEVel][:AMPLitude] <n>Set fixed source 2 amplitude

Parameters   <n> =   -30 to 30   Set source amplitude (volts)
                           DEFault   0 V
                           MINimum   -30 V
                           MAXimum   30 V

Query   :VOLTage?   Query programmed source amplitude
                   :VOLTage? DEFault   Query *RST default amplitude
                   :VOLTage? MINimum   Query lowest allowable amplitude
                   :VOLTage? MAXimum   Query highest allowable amplitude

Description   This command is used to immediately update the amplitude of a fixed source. This command is not valid if the list or sweep mode is selected.

NOTE   *The sourcing :MODE command is used to select a fixed source. See "Select sourcing mode," page Section 16-40.*

If a manual source range is presently selected, then the specified amplitude cannot exceed that range. For example, if the source is on the 10 V range (autorange disabled), you will not be able to set the source amplitude to 20 V. In autorange, the amplitude can be set to any level that is within the capabilities of the source.

The MINimum and MAXimum parameters are only valid if the 30 V source range is presently selected. Sending the MINimum or MAXimum parameters on the 10 V source range will generate error -221 (Setting Conflict).

TRIGgered[:AMPLitude] <n>

:SOURce[1]:VOLTage[:LEVel]:TRIGgered[:AMPLitude] <n>Set fixed source 1 amplitude when triggered
:SOURce2:VOLTage[:LEVel]:TRIGgered[:AMPLitude] <n>Set fixed source 2 amplitude when triggered

Parameters   <n> =   -30 to 30   Set source amplitude (volts)
                           DEFault   0 V
                           MINimum   -30 V
                           MAXimum   +30 V

Query	:TRIGgered?	Query triggered amplitude for fixed source
	:TRIGgered? DEFault	Query *RST default amplitude
	:TRIGgered? MINimum	Query lowest allowable amplitude
	:TRIGgered? MAXimum	Query highest allowable amplitude

Description   These commands perform the same as the [:IMMediate][:AMPLitude] commands except that the source amplitude is not updated immediately.

With this command, the amplitude is updated when the Model 6482 is triggered to perform a source and measure operation. For example, if the instrument is waiting in the trigger layer for an external trigger, the amplitude of the source will not update until that external trigger is received by the Model 6482. See Section 8, "Triggering," for details on trigger model operation.

The MINimum and MAXimum parameters are only valid if the 30 V source range is presently selected. Sending the MINimum or MAXimum parameters on the 10 V source range will generate error -221 (Setting Conflict).

Set delay

NOTE   All SOURx:DEL commands are global and affect both simultaneously.

DELay <n>

	:SOURce[1]:DELay <n>	Manually set source delay
	:SOURce2:DELay <n>	Manually set source delay

Parameters	<n> =	0 to 999.998	Specify delay in seconds
		MINimum	0 seconds
		MAXimum	999.998 seconds
		DEFault	0.001

Query	:DELay?	Query delay
	:DELay? DEFault	Query *RST default delay
	:DELay? MINimum	Query lowest allowable delay
	:DELay? MAXimum	Query highest allowable delay

Description   This command is used to manually set a delay (settling time) for the source. After the programmed source is turned on, this delay occurs to allow the source level to settle before a measurement is taken. Note that this delay is the same for both channels.

Do not confuse this source delay with the trigger delay. The source delay is part of the device action (SDM cycle) while the trigger delay occurs before the device action. See Section 8, "Front-panel operation of the trigger model," for more information.

Autodelay can instead be used to automatically set the source delay. See the next command.

AUTO <b>

	:SOURce[1]:DELay:AUTO <b>	Enable or disable automatic source delay
	:SOURce2:DELay:AUTO <b>	Enable or disable automatic source delay

Parameters	<b> =	0 or OFF	Disable autodelay
		1 or ON	Enable autodelay

Query	:AUTO?	Query state of autodelay

Description   These commands are used to enable or disable autodelay. When enabled, the instrument will automatically select a delay period. *RST and SYST:PRES default is OFF. Note that this delay is the same for both channels.

Select ground connect mode

GCONnect <b>

    :SOURce[1]:GCONnect <b>   Enable or disable source 1 ground connect
    :SOURce2:GCONnect <b>   Enable or disable source 2 ground connect

Query   :GCONnect?   Query state of ground connect

Description   These commands are used to enable or disable the ground connect mode of the channel 1 and channel to 2 sources. When ground connect is enabled, the SOURCE OUTPUT terminal is connected to chassis ground. When ground connect is disabled, SOURCE is left floating.

Configure sweeps

There are two methods to configure the start and stop levels of a sweep. You can use either the :STARt and :STOP commands or you can use the :CENTer and :SPAN commands. See "Sweep and list program examples" later in this section.

NOTE   *In order to run a sweep, the selected source must be in the sweep sourcing mode and the trigger count should be the same as the number of source and measure points in the sweep.*

Use the VOLTage:MODE command to select the SWEep sourcing mode. See "Select sourcing mode," page Section 16-40. The trigger count is set using the TRIGger:COUNt command. See "Trigger subsystem," page Section 16-58.

RANGing <name>

    :SOURce[1]:SWEep:RANGing <name>Select source 1 ranging mode
    :SOURce2:SWEep:RANGing <name>Select source 2 ranging mode

Parameters   <name> = BEST   Use the best fixed mode
                          AUTO   Use the most sensitive source range for
                                        each sweep level
                          FIXed   Use the present source range for the
                                        entire sweep

Query   :RANGing?   Query source ranging mode

Description   These commands are used to select the source ranging mode for sweeps using the channel 1 and channel 2 voltage sources. With BEST selected, the Model 6482 will select a single fixed source range that will accommodate all the source levels in the sweep. For front-panel operation, this is the BEST FIXED option.

With AUTO selected, the Model 6482 will automatically go to the most sensitive source range for each source level in the sweep. For front-panel operation, this is the AUTO RANGE option.

With FIXed selected, the source remains on the range that it is presently on when the sweep is started. For sweep points that exceed the source range capability, the source will output the maximum level for that range. For front-panel operation, this is the FIXED option.

SPACing <name>

:SOURce[1]:SWEep:SPACing <name>Select scale for source 1 sweep
:SOURce2:SWEep:SPACing <name>Select scale for source 2 sweep

Parameters   <name>   = LINear   Linear scale
                          LOGarithmic   Logarithmic scale

Query   :SPACing?   Query scale for sweep

Description   These commands are used to select the scale for the sweep. With LINear selected, the source and measure points in the sweep will be performed on a linear scale. With LOGarithmic selected, the source and measure points will be performed on a logarithmic scale.

STARt <n>
STOP <n>

:SOURce[1]:VOLTage:STARt <n>Specify source 1 start voltage level
:SOURce[1]:VOLTage:STOP <n>Specify source 1 stop voltage level
:SOURce2:VOLTage:STARt <n>Specify source 2 start voltage level
:SOURce2:VOLTage:STOP <n> Specify source 2 stop voltage level

Parameters   <n> =   -30 to 30   Set source level (volts)
                         DEFault   0 V
                         MINimum   -30 V
                         MAXimum   +30 V

Query   :STARt?   Query start level for sweep
                   :STARt? DEFault   Query *RST default level
                   :STARt? MINimum   Query lowest allowable level
                   :STARt? MAXimum   Query highest allowable level

                   :STOP?   Query start level for sweep
                   :STOP? DEFault   Query *RST default level
                   :STOP? MINimum   Query lowest allowable level
                   :STOP? MAXimum   Query highest allowable level

Description   These commands are used to specify the start and stop levels for a sweep. If using a fixed (manual) source range, the sweep will be performed using a source range that will accommodate all source values (best fixed range). You can use source autorange if sweeping through one or more source ranges.

When the sweep is started, the source will output the specified start level and, after the delay period of the SDM cycle, a measurement is performed.

The sweep continues until the source outputs the specified stop level. At this level, the instrument again performs another measurement (after the SDM delay) and then stops the sweep.

The source and measure points in a sweep can be set by specifying a step size, or by specifying the number of source and measure points in the sweep. Refer to "STEP <n>," page Section 16-46, and "POINts <n>," page Section 16-47.

:STARt and :STOP are coupled to :CENTer and :SPAN. Thus, when start and stop values are changed, the values for center and span are affected as follows:

Center = (Start + Stop) / 2
Span = Stop - Start

CENTer <n>
SPAN <n>

:SOURce[1]:VOLTage:CENTer <n>Specify center point of source 1 sweep
:SOURce[1]:VOLTage:SPAN <n>Specify span of source 1 sweep
:SOURce2:VOLTage:CENTer <n>Specify center point of source 2 sweep
:SOURce2:VOLTage:SPAN <n> Specify span of source 2 sweep

Parameters   <n> =

-30 to 30	Set SPAN source level (volts)
-30 to 30	Set CENT source level (volts)
DEFault	0 V
MINimum	-30 V (SPAN), -30 V (CENT)
MAXimum	+30 V (SPAN), +30 (CENT)

Query

:CENTer?	Query center point for sweep
:CENTer? DEFault	Query *RST default level
:CENTer? MINimum	Query lowest allowable level
:CENTER? MAXimum	Query highest allowable level
:SPAN?	Query span for sweep
:SPAN? DEFault	Query *RST default level
:SPAN? MINimum	Query lowest allowable level
:SPAN? MAXimum	Query highest allowable level

Description   A sweep can be configured by specifying center and span parameters. By specifying a center point, you can sweep through the operating point of a device. The span determines the sweep width with the operating point at the center of the sweep.

For example, assume you are testing a device that operates at 10 V, and you want to sweep from 8 to 12 volts. To do this, you would specify the center to be 10 V and the span to be 4 volts (12 - 8).

Use the :STEP or :POINts command to specify the number of source and measure points in the sweep.

:CENTer and :SPAN are coupled to :STARt and :STOP. Thus, when center and span values are changed, the values for start and stop are affected as follows:
Start = Center - (Span / 2)
Stop = Center + (Span / 2)

STEP <n>

:SOURce[1]:VOLTage:STEP <n>Specify step size (source 1 sweep)
:SOURce2:VOLTage:STEP <n> Specify step size (source 2 sweep)

Parameters   <n> =

-30 to 30	Set source level (volts)
DEFault	0 V
MINimum	-30 V
MAXimum	+30 V

Query

:STEP	Query step size for sweep
:STEP? DEFault	Query *RST default level
:STEP? MINimum	Query lowest allowable level
:STEP? MAXimum	Query highest allowable level

Description   These commands are used to specify a step size for a linear sweep. When the sweep is started, the source level changes in equal steps from the start level to the stop level. A measurement is performed at each source step (including the start and stop levels).

NOTE   *This command cannot be used for a logarithmic sweep. Use the :POINts command to set the source and measure points for a logarithmic sweep.*

To avoid a setting conflicts error, make sure the step size is greater than the start value and less than the stop value.

The number of source and measure points in a linear sweep can be calculated as follows:

Points = [(Stop - Start) / Step] + 1
Points = (Span / Step) + 1

An alternate way to set the source and measure points in a linear sweep is to simply specify the number of source and measure points in the sweep using the :POINts command.

Note that the :STEP and :POINts commands are coupled. Changing the step size also changes the number of source and measure points. Conversely, changing the number of source and measure points changes the step size.

POINts <n>

:SOURce[1]:SWEep:POINts <n>   Set number of points for source 1
:SOURce2:SWEep:POINts <n>   Set number of points for source 2

Parameters	<n> =	1 to 3000	Specify number of source-measure points
		MINimum	1
		MAXimum	3000
		DEFault	3000

Query	:POINts?	Query number of sweep points
	:POINts? DEFault	Query *RST default number of sweep points
	:POINts? MINimum	Query lowest allowable number of sweep points
	:POINts? MAXimum	Query highest allowable number of sweep points

Description   The :POINts command specifies the total number of source and measure points in a sweep. For a linear sweep, the source and measure points are equally spaced (stepped) between the start level and the stop level. For a logarithmic sweep, the source and measure points are equally spaced on a logarithmic scale. Note that the start and stop levels are source and measure points.

Step size for a linear sweep can be calculated as follows:

Step Size = (Stop - Start) / (Points - 1)
Step Size = Span / (Points -1)

Step size for a logarithmic sweep can be calculated as follows:

$$\text{Log Step Size} = \frac{\log 10\,(\text{Stop}) - \log 10\,(\text{Start})}{\text{Points - 1}}$$

An alternate way to set the source and measure points in a sweep is to specify the step size using the :STEP command.

Note that the :POINts and :STEP commands are coupled. Changing the number of source and measure points also changes the step size. Conversely, changing the step size changes the number of source and measure points.

DIRection <name>

:SOURce[1]:SWEep:DIRection <name>Set direction of source 1 sweep
:SOURce2:SWEep:DIRection <name>Set direction of source 2 sweep

Parameters   <name> = UP   Run sweep from start to stop
                            DOWN   Run sweep from stop to start

Query   :DIRection?   Query direction of sweep

Description   Normally, a sweep is run from the start level to the stop level. The :STARt and :STOP, or :CENTer and :SPAN commands are used to set these levels.

This command lets you change the execution direction of the sweep. With DOWN selected, the sweep will begin at the stop level and end at the start level. Selecting UP restores sweep operation to the normal start to stop direction.

Configure list

VOLTage <NRf list>

:SOURce[1]:LIST:VOLTage <NRf list>Define source 1 list
:SOURce2:LIST:VOLTage <NRf list>Define source 2 list

Parameters   <NRf list> = NRf, NRf … NRf
                   NRf =   -30 to 30   Source value (volts)

Query   :VOLTage?   Query source list

Description   These commands are used to define a list of source values (up to 100) for the list sourcing mode of operation. When operation is started, the instrument will sequentially source each voltage value in the list. A current measurement is performed at each source level.

The following command shows the proper format for defining a list using source values of 10 V, 13 V, and 5 V:

:SOURce[1]:LIST:VOLTage 10, 13, 5

If using manual source ranging, you can use autorange for source values that are not within a single range.

NOTE   *In order to execute a source list, the selected source must be in the list sourcing mode, and the product of the arm and trigger count should be at least the same as the number of source points in the list.*

Use the VOLTage:MODE command to select the LIST sourcing mode. See "Select sourcing mode," page Section 16-40. The trigger count is set using the TRIGger:COUNt command. See "Trigger subsystem," page Section 16-58.

APPend <NRf list>

:SOURce[1]:LIST:VOLTage:APPend <NRf list>Add value(s) to source 1 list
:SOURce2:LIST:VOLTage:APPend <NRf list>Add value(s) to source 2 list

Parameters   <NRf list> = NRf, NRf … NRf
                   NRf =   -30 to 30   Source value (volts)

Description   These commands are used to add one or more values (up to 100) to a source list that already exists. The source values are appended to the end of the list. By using multiple appended lists, up to 3000 points can be in a list.

POINts?

:SOURce[1]:LIST:VOLTage:POINts?Query length of source 1 list
:SOURce2:LIST:VOLTage:POINts?Query length of source 2 list

Description   These commands are used to determine the length of the specified source list.
The response message indicates the number of source values in the list.

Sweep and list program examples

Linear voltage sweep

Linear source 1 voltage sweep from 1 V to 10 V in 1 V increments:

```
*RST
SOUR1:SWE:SPAC LIN
SOUR1:VOLT:STAR 1
SOUR1:VOLT:STOP 10
SOUR1:VOLT:STEP 1
SOUR1:SWE:POIN? (returns 10)
TRIG:COUN 10
SOUR1:VOLT:MODE SWE
OUTP1 ON
INIT
```

List sweep

The previous linear sweep can instead be performed using a voltage list as follows:

```
*RST
SOUR1:LIST:VOLT 1,2,3,4,5,6,7,8,9,10
SOUR1:LIST:VOLT:POIN? (returns 10)
TRIG:COUN 10
SOUR1:VOLT:MODE LIST
OUTP1 ON
INIT
```

Logarithmic sweep

Logarithmic sweep from 1 V to 10 V in 5 points:

```
*RST
SOUR1:SWE:SPAC LOG
SOUR1:VOLT:STAR 1
SOUR1:VOLT:STOP 10
SOUR1:SWE:POIN 5
TRIG:COUN 5
SOUR1:VOLT:MODE SWE
OUTP1 ON
INIT
```

To determine the source values that will be generated:

Start: 1 Log_{10}(Start): 1
Stop: 10 Log_{10}(Stop): 2

$$LogStep = (Log_{10}(Start) - Log_{10}(Stop)) / (SWE:POIN - 1)$$
$$= (1-0) / (5 - 1)$$
$$= 1/4$$
$$= 0.25$$

Now add the LogStep value to Log_{10}(Start) and to each subsequent result. This will create a list of Log_{10} values. Next take the anti-log of each Log_{10} value to get the actual sweep values:

Point	Logarithmic step	Sweep Value (V)
1	0	1.0000
2	0.25	1.7783
3	0.50	3.1623
4	0.75	5.6234
5	1.0	10.0000

STATus subsystem

The STATus subsystem is used to control the status registers of the Model 6482. The commands in this subsystem are summarized in Table 16-7.

NOTE   *These registers and the overall status structure are fully explained in Section 13.*

Read event registers

[:EVENt]?

:STATus:MEASurement[:EVENt]?Read Measurement Event Register
:STATus:QUEStionable[:EVENt]?Read Questionable Event Register
:STATus:OPERation[:EVENt]?   Read Operation Event Register

Description   These query commands are used to read the contents of the status event registers. After sending one of these commands and addressing the Model 6482 to talk, a value is sent to the computer. This value indicates which bits in the appropriate register are set.

Program event enable registers

ENABle <NDN> or <NRf>

:STATus:MEASurement:ENABle <NDN> or <NRf>Program Measurement Event Enable Register
:STATus:QUEStionable:ENABle <NDN> or <NRf>Program Questionable Event Enable Register
:STATus:OPERation:ENABle <NDN> or <NRf>Program Operation Event Enable Register

Parameters   <NDN> =   #Bxx...x   Binary format (each x = 1 or 0)
                            #Hx   Hexadecimal format (x = 0 to FFFF)
                            #Qx   Octal format (x = 0 to 177777)
                  <NRf> =   0 to 65535   Decimal format

Query   :ENABle?   Read an enable register

Description   These commands are used to program the enable registers of the status structure. The binary equivalent of the parameter value that is sent determines which bits in the register gets set. See Section 13 for details.

Read condition registers

CONDition?

:STATus:MEASurement:CONDition?Read Measurement Condition
:STATus:QUEStionable:CONDition?Read Questionable Register
:STATus:OPERation:CONDition?Read Operation Condition

Description   These query commands are used to read the contents of the condition registers.

Select default conditions

PRESet

:STATus:PRESet   Return registers to default conditions

Description   When this command is sent, the following SCPI event registers are cleared to zero (0):

1. Operation Event Enable Register.
2. Event Enable Register.
3. Measurement Event Enable Register.

NOTE   *The Standard Event Register is not affected by this command.*

Error queue

[:NEXT]?

:STATus:QUEue[:NEXT]?   Read error queue

Description   As error and status messages occur, they are placed into the error queue. This query command is used to read those messages. See Appendix A for a list of messages.

NOTE   *The :STATus:QUEue[:NEXT]? query command performs the same function as the :SYSTem:ERRor? query command. See "SYSTem subsystem," page Section 16-52.*

CLEar

:STATus:QUEue:CLEar   Clear error queue

Description   This action command is used to clear the error queue of messages.

ENABle <list>

:STATus:QUEue:ENABle <list>   Enable messages for error queue

Parameters   <list> = (numlist)

where numlist is a specified list of messages that you wish to enable for the Error.

Query   :ENABle?   Query list of enabled messages

Description   On power-up, all error messages are enabled and will go into the error queue as they occur. Status messages are not enabled and will not go into the queue. This command is used to specify which messages you want enabled. Messages not specified will be disabled and prevented from entering the queue.

DISable <list>

:STATus:QUEue:DISable <list>   Disable messages for error queue

Parameters   <list> = (numlist)

where numlist is a specified list of messages that you wish to disable for the error queue.

Query   :DISable?   Query list of disabled messages

Description On power-up, all error messages are enabled and will go into the error queue as they occur. Status messages are not enabled and will not go into the queue. This command is used to specify which messages you want disabled. Disabled messages are prevented from going into the error queue.

SYSTem subsystem

The SYSTem subsystem contains miscellaneous commands that are summarized in Table 16-8.

Default conditions

PRESet

:SYSTem:PRESet   Return to :SYSTem:PRESet defaults

Description This command returns the instrument to states optimized for front-panel operation. :SYSTem:PRESet defaults are listed in the SCPI tables (Table 16-1 through Table 16-10).

POSetup

:SYSTem:POSetup <name>   Program power-on defaults

Parameters   <name> =   RST   Power-up to *RST defaults
                           PRESet   Power-up to :SYSTem:PRESet defaults
                           SAV0   Power-up to setup stored at memory location 0
                           SAV1   Power-up to setup stored at memory location 1
                           SAV2   Power-up to setup stored at memory location 2
                           SAV3   Power-up to setup stored at memory location 3
                           SAV4   Power-up to setup stored at memory location 4

Query   :POSetup?   Query power-on setup

Description This command is used to select the power-on defaults. With RST selected, the instrument powers up to the *RST default conditions. With PRES selected, the instrument powers up to the :SYSTem:PRESet default conditions. Default conditions are listed in the SCPI tables (Table 16-1 through Table 16-10).

With the SAV0-4 parameters specified, the instrument powers-on to the setup that is saved in the specified location using the *SAV command.

Control autozero

STATe <name>

:SYSTem:AZERo:STATe <name>Control autozero

Parameters   <name> = ON   Enable autozero
                           OFF   Disable autozero

Query   :STATe?   Query state of autozero

Description   This command is used to enable or disable autozero, or to force an immediate one-time autozero update if autozero is disabled. When autozero is enabled, accuracy is optimized. When autozero is disabled, speed is increased at the expense of accuracy. Note that both channels are affected by this command.

Select power line frequency setting

LFRequency <freq>

:SYSTem:LFRequency <freq>   Select line frequency

Parameters   <freq> =   50   50Hz setting
                             60   60Hz setting

Query   :LFRequency?   Query line frequency selection.

Description   Use this command to manually select the line frequency setting (50 or 60Hz).

Error queue

NOTE   *See Section 13 for details on the error queue.*

[:NEXT]?

:SYSTem:ERRor[:NEXT]?   Read oldest error (code and message)

Description   As error and status messages occur, they are placed in the error queue. The error queue is a first-in, first-out (FIFO) register that can hold up to 10 messages. After sending this command and addressing the Model 6482 to talk, the oldest message is sent to the computer and is then removed from the queue.

NOTE   *The :STATus:Queue? command performs the same function as :SYSTem:ERRor[:NEXT]?. See "STATus subsystem," page Section 16-50.*

ALL?

:SYSTem:ERRor:ALL?   Read all errors (codes and messages)

Description   This query command is similar to the [:NEXT]? command except that all messages in the error queue are sent to the computer when the Model 6482 is addressed to talk. All messages are removed from the queue.

COUNt?

:SYSTem:ERRor:COUNt?   Return the number of errors

Description   After sending this command and addressing the Model 6482 to talk, a decimal number will be sent to the computer. That is the number of messages in the error queue.

CODE[:NEXT]?

:SYSTem:ERRor:CODE[:NEXT]?Read oldest error (code only)

Description   This command is identical to the [:NEXT]? command, except only the code is returned. The message itself is not returned. The error is cleared from the queue.

CODE:ALL?

SYSTem:ERRor:CODE:ALL?   Read all errors (codes only)

Description   This query command is identical to the :ALL? command, except only the codes are returned. The actual messages are not returned. All errors are cleared from the queue.

CLEar

:SYSTem:CLEar   Clear error queue

Description   This action command is used to clear the error queue of messages.

Simulate key presses

KEY

:SYSTem:KEY <NRf>   Simulate key-press

Parameters   <NRf> =   1   RANGE ▲ key
                           2   EDIT ▼ key
                           3   EDIT ◄ key
                           4   MENU key
                           5   DELTA key
                           6   FILTER key
                           7   SPEED key
                           8   CHANNEL SELECT key
                           9   AUTO key
                           10   EDIT ▶ key
                           11   EXIT key
                           12   SRC1 key
                           13   LIMITS key
                           14   STORE key
                           15   MSR1 key
                           16   DISPLAY TOGGLE key
                           17   RANGE ▼ key
                           18   ENTER key
                           19   SRC2 key
                           20   TRIG key
                           21   RECALL key
                           22   MSR2 key
                           23   LOCAL key
                           24   ON/OFF key
                           25   -----
                           26   EDIT ▲ key
                           27   SWEEP key
                           28   CONFIG key
                           29   RATIO
                           30   REL key
                           31   DIGITS key

Query   :KEY?   Query last pressed key.

Description   This command is used to simulate front-panel key presses. For example, to select the channel 1 measurement function (MSR1), you can send the following command to simulate pressing the MSR1 key:

:syst:key 15

The parameter listing above provides the key-press code in numeric order.

The queue for the :KEY? query command can only hold one key-press. When :KEY? is sent over the bus, and the Model 6482 is addressed to talk, the key-press code number for the last key pressed (either physically or with :KEY) is sent to the computer.

The key-press code number for the last key pressed (either physically or with :key) is sent to the computer.

Read version of SCPI standard

VERSion?

:SYSTem:VERSion?   Read SCPI version

Description   This query command is used to read the version of the SCPI standard being used by the Model 6482. Example code:

1996.0

The above response message indicates the version of the SCPI standard.

RS-232 interface

NOTE   *The following commands are intended for use over the RS-232 interface, but they can also be used over the GPIB.*

LOCal

:SYSTem:LOCal   Take Model 6482 out of remote

Description   Normally, during RS-232 communications, front-panel keys are operational. However, the user may wish to lock out front-panel keys during RS-232 communications. See RWLock.

This action command is used to remove the Model 6482 from the remote state and enables the operation of front-panel keys in a manner similar to the GPIB GTL command.

REMote

:SYSTem:REMote   Place the Model 6482 in remote

Description   This action command is used to place the Model 6482 in the remote state in a manner similar to the GPIB REN command. In remote, the front-panel keys will be locked out if local lockout is asserted. See "RWLock," page Section 16-55.

RWLock

:SYSTem:RWLock   Disable or enable front-panel keys

Description   This command is used to enable local lockout in a manner similar to the GPIB LLO command (see Section 12). When enabled, the front-panel keys except OUTPUT OFF are locked out (not operational) when the instrument is in remote.

For additional information, see "REMote". When disabled, the front panel LOCAL key is operational in remote.

Removing the instrument from remote (:LOCal) restores front-panel keys operation.

Reset timestamp

RESet

:SYSTem:TIME:RESet   Reset timestamp

Description   This action command is used to reset the absolute timestamp to 0 seconds. The timestamp also resets to 0 seconds every time the Model 6482 is turned on.

TRACe subsystem

The commands in this subsystem are used to configure and control data storage into the   buffer. The commands are summarized in Table 16-9.

:TRACe|:DATA

The bar (|) indicates that :TRACe or :DATA can be used as the root command for this subsystem. From this point on, the documentation in this manual uses :TRACe. If you prefer to use :DATA, simply replace all the :TRACe command words with :DATA.

Read and clear buffer

DATA?

:TRACe:DATA?   Read contents of buffer

Description   When this command is sent and the Model 6482 is addressed to talk, all the readings stored in the data store are sent to the computer.

The format used to send readings over the bus is controlled from the :FORMat subsystem.

NOTE   *See Appendix B for a detailed explanation on how data flows through the various operation blocks of the* Model 6482*. It clarifies the types of readings that are acquired by the various commands to read data.*

CLEar

:TRACe:CLEar   Clear buffer

Description   This action command is used to clear the buffer of readings. If you do not clear the buffer, a subsequent store will overwrite the old readings.

Configure and control buffer

FREE?

	:TRACe:FREE?	Read status of memory

Description   This command is used to read the status of storage memory. After sending this command and addressing the Model 6482 to talk, two values separated by commas are sent to the computer. The first value indicates how many bytes of memory are available, and the second value indicates how many bytes are reserved to store readings.

POINts <n>

	:TRACe:POINts <n>	Specify buffer size

Parameters   <n> =
	1 to 3000	Specify buffer size
	MINimum	1
	MAXimum	3000
	DEFault	100

Query
:POINts?	Query buffer size
:POINts? MINimum	Query smallest allowable buffer size
:POINts? MAXimum	Query largest allowable buffer size
:POINts? DEFault	Query *RST default buffer size

Description   This command is used to specify the size of the buffer.

ACTual?

:TRACe:POINts:ACTual?	Query number of stored readings

Description   This query command is used to determine how many stored readings are in the buffer. After sending this command and addressing the instrument to talk, the number of readings stored in the buffer will be sent to the computer.

CONTrol <name>

:TRACe:FEED:CONTrol <name>	Start or stop buffer

Parameters   <name> =
	NEXT	Fills buffer and stops
	NEVer	Disables buffer storage

Query   :CONTrol?   Query buffer control

Description   This command is used to select the buffer control. When NEXT is selected, the asterisk (*) indicator turns on to indicate that the buffer is enabled. The storage process starts when Model 6482 is taken out of idle to perform source-measure operations.

After the buffer stores the specified number of reading arrays (as set by the :POINts command), the asterisk annunciator turns off to indicate that storage is done.

With NEVer selected, storage into the buffer is disabled.

Select timestamp format

FORMat <name>

:TRACe:TSTamp:FORMat <name>Select timestamp format

| **Parameters** | <name> = | ABSolute | Reference to first buffer reading |
| | | DELTa | Time between buffer readings |

Query   :FORMat?   Query timestamp format

Description   This command is used to select the timestamp format for buffer readings. With ABSolute selected, each timestamp is referenced to the first reading stored in the buffer. With DELTa selected, timestamps provide the time between each buffer reading.

Trigger subsystem

The Trigger subsystem is made up of a series of commands and subsystems to configure the Trigger Model. These commands and subsystems are summarized in Table 16-10.

NOTE   *See Section 8 for more details on triggering and the trigger model.*

Initiate source/measure cycle

INITiate

:INITiate[:IMMediate]   Take Model 6482 out of idle state

Description   This command is used to initiate source and measure operation by taking the Model 6482 out of idle. The :READ? and :MEASure? commands also perform an initiation.

Command processing will halt until the trigger model returns to the idle state, with the following exceptions:

- ABORt
- SDC
- DCL
- IFC
- SYSTem:PRESet, *RST, or *RCL
- *TRG or GET with TRIG:SOUR BUS

If any other command is sent after an INIT, only SDC, IFC, and DCL will work.

Abort source/measure cycle

ABORt

:ABORt   Abort operation

Description   When this action command is sent, the Model 6482 aborts operation and returns to the idle state.

A faster way to return to idle is to use the DCL or SDC command.

Program trigger model

COUNt <n>

:ARM[:SEQuence[1]][LAYer[1]]:COUNt <n> Set arm count
:TRIGger[:SEQuence[1]]:COUNt <n> Set trigger count

Parameters   <n> =   1 to 3000   Specify count (see NOTE)
                              DEFault   Sets count to 1
                              MINimum   Sets count to 1
                              MAXimum   See NOTE
                              INFinite   (ARM:COUNt only)

NOTE   *The product of arm count and trigger count cannot exceed 3000.*

Query   :COUNt?   Queries programmed count
              :COUNt? DEFault   Queries *RST default count
              :COUNt? MINimum   Queries lowest allowable count
              :COUNt? MAXimum   Queries largest allowable count

Description   This command is used to specify how many times an operation is performed in the specified layer of the trigger model.

For example, assume the arm count is set to 2 and the trigger counter is set to 10, the Model 6482 is configured to perform 10 source-measure operations twice for a total of 20 source-measure operations.

The product of the arm count and trigger count cannot exceed 3000. If, for example, the arm count is 2, then the maximum trigger count is 1250.

NOTE   *INFinite can be used only with ARM:COUNt, and FETCh?. READ?, MEAS?, CALC1:DATA? through CALC7:DATA? cannot be used with infinite arm count. Only INIT will start measurements, and only interlock, over-temperature, SDC, DCL, or ABORt should be used to stop the sweep.*

ARM:COUNt INFinite can be used for repetitive source waveforms or for long tests where only the last reading is important. For example, the limits could be used to drive the interlock to abort a test when some condition is met. :DATA? would then give the answer to the test.

DELay <n>

:TRIGger[:SEQuence[1]]:DELay <n> Set trigger layer delay

Parameters   <n> =   0 to 999.9999   Specify delay in seconds
                              DEFault   0 second delay
                              MINimum   0 second delay
                              MAXimum   999.9999 second delay

Query   :DELay?   Query the programmed delay
              :DELay? DEFault   Query the *RST default delay
              :DELay? MINimum   Query the lowest allowable delay
              :DELay? MAXimum   Query the largest allowable delay

Description   The delay is used to delay operation in the trigger layer. After the programmed trigger event occurs, the instrument waits until the delay period expires before performing the device action.

SOURce <name>

:ARM[:SEQuence[1]][LAYer[1]]:SOURce <name>Specify arm event control source
:TRIGger[:SEQuence[1]]:SOURce <name>Specify trigger event control source

Parameters   <name> =   IMMediate   Pass operation through immediately
                              TLINk   Select Trigger Link trigger as event
                              TIMer   Select timer as event
                              MANual   Select manual event
                              BUS   Select bus trigger as event
                              NSTest   Select low SOT pulse as event
                              PSTest   Select high SOT pulse as event
                              BSTest   Select high or low SOT pulse as event

NOTE   Only IMMediate and TLINk are available as trigger layer control sources.

Query   :SOURce?   Query programmed control source

Description   These commands are used to select the event control source. With IMMediate selected, operation immediately continues.

A specific event can be used to control operation. With TLINk selected, operation continues when a trigger pulse is received using the trigger link.

NOTE   The following control sources are not available for the trigger layer. For more information on these control sources, see "Arm layer," page Section 8-16.

With TIMer selected, the event occurs at the beginning of the timer interval, and every time it times out. For example, if the timer is programmed for a 30 second interval, the first pass through the control source occurs immediately. Subsequent arm events will then occur every 30 seconds. The interval for the timer is set using the :TIMer command.

With MANual selected, the event occurs when the TRIG key is pressed.

With BUS selected, the event occurs when a GET or *TRG command is sent over the bus.

With NSTest selected, the event occurs when the SOT (start-of-test) low pulse is received. This is used for limit testing.

With PSTest selected, the event occurs when SOT (start-of-test) high pulse is received. This is used for limit testing.

With BSTest selected, the event occurs when SOT (start-of-test) high or low pulse is received. This is used for limit testing.

TIMer <n>

:ARM[:SEQuence[1]][:LAYer[1]]:TIMer <n>Set interval for arm layer timer

Parameters   <n> =   0.001 to 9999.999Specify timer interval in seconds
                          10000.00 to 99999.99Specify timer interval in seconds

Query   :TIMer?   Query programmed timer interval

Description   These commands are used to set the interval for the timer. Note that the timer is in effect only if the timer is the selected control source.

DIRection <name>

:ARM[:SEQuence[1]][LAYer[1]][:TCONfigure]:DIRection <name>Control arm bypass
:TRIGger[:SEQuence[1]][:TCONfigure]:DIRection <name>Control trigger bypass

Parameters   <name> =   SOURce   Enable control source bypass
                              ACCeptor   Disable control source bypass

Query   :DIRection?   Query state of bypass

Description   This command is used to enable (SOURce) or disable (ACCeptor) control source bypass. When enabled, operation will loop around the control source on the first pass in the layer. After that, repeat passes in the layer are held up and will wait for the programmed control source event.

INPut <event list>

:TRIGger[:SEQuence[1]][:TCONfigure][:ASYNchronous]:INPut <event list>Enable event detectors

Parameters   <event list> =SOURceEnable Source Event Detector
                              DELayEnable Delay Event Detector
                              SENSeEnable Measure Event Detector
                              NONE   Disable all event detectors in Trigger Layer

NOTE   *Each event in the list must be separated by a comma (i.e. trigger:input source, delay, sense).*

Query   :INPut?   Query enabled event detectors in Trigger Layer

Description   When TLINk is the selected Trigger Layer control source, and an event detector in the Trigger Layer is enabled, operation will hold up at that detector until an input trigger is received using the trigger link. When the event detector is disabled, operation will not hold up. It continues and performs the appropriate action.

A Trigger Layer event detector is enabled by including the parameter name in the event list for the INPut command. For example, to enable the Source Event Detector and Measure Event Detector, send the following command:

:TRIGger:INPut SOURce, SENSe

The Delay Event Detector will be disabled since the DELay parameter is not included in the above event list.

NOTE   *To disable all the Trigger Layer event detectors, the NONE parameter must be sent alone (i.e. trigger:input none). If it is listed with any of the other parameters, NONE will be ignored.*

ILINe <NRf>

:ARM[:SEQuence[1]][LAYer[1]][:TCONfigure]:ILINe <NRf>Select input line; arm layer
:TRIGger[:SEQuence[1]][:TCONfigure]:ILINe <NRf>Select input line; trigger layer

Parameters   <NRf> =   1   Line #1
                              2   Line #2
                              3   Line #3
                              4   Line #4
                              5   Line #5
                              6   Line #6

Query   :ILINe?   Query input trigger line

Description   This command is used to select input lines for the Trigger Link. For normal operation, Trigger Link input and output ("OLINe") should not share the same line.

OLINe <NRf>

:ARM[:SEQuence[1]][LAYer[1]][:TCONfigure]:OLINe <NRf>Select output line; arm layer
:TRIGger[:SEQuence[1]][:TCONfigure]:OLINe <NRf>Select output line; trigger layer

Parameters   <NRf> =   1   Line #1
                               2   Line #2
                               3   Line #3
                               4   Line #4
                               5   Line #5
                               6   Line #6

Query   :OLINe?   Query output trigger line

Description   This command is used to select output lines for the Trigger Link. For normal operation, Trigger Link input and output ("ILINe") should not share the same line.

OUTPut <event list>

:ARM[:SEQuence[1]][LAYer[1]][:TCONfigure]:OUTPut <event list>Arm layer event
:TRIGger[:SEQuence[1]][:TCONfigure]:OUTPut <event list>Trigger layer events

Parameters   Arm Layer Trigger
                   <event list> = TRIGger   Trigger on exiting trigger layer
                                  NONE   Disable arm layer output trigger
                   Trigger Layer Triggers
                   <event list> = SOURce   Output trigger after source level is set
                                  DELay   Output trigger after delay period
                                  SENSe   Output Trigger after measurement
                                  NONE   Disable trigger layer triggers

NOTE   *Each event in the list must be separated by a comma (for example, :arm:output source, delay, sense).*

Query   :OUTPut?   Query output trigger event(s)

Description   This command is used to specify when trigger pulses occur on the specified output trigger line of the Trigger Link (:OLINe).

Arm Layer Trigger: With TRIGger selected, an output trigger will occur when exiting the trigger layer. With NONE selected, the arm layer output trigger is disabled.

Trigger Layer Triggers: You can specify from one to all three events. Each event in the list must be separated by a comma (,).

The SOURce, DELay and MEASure events refer to the source-delay-measure (SDM) cycle. This is the Device Action in the Trigger Model. See Figure 8-7 and Figure 8-8. With SOURce specified, an output trigger occurs after the source is set. With DELay specified, an output trigger occurs after the delay period. With MEASure specified, an output trigger occurs after the measurement.

NOTE   *When disabling triggers, the NONE parameter must be sent alone (for example, trig:outp none). If it is listed with any of the other event parameters, NONE will be ignored.*

In this section:

Performance verification

WARNING   ***The information in this section is intended only for qualified service personnel. Some of the procedures may expose you to hazardous voltages that could result in personal injury or death. Do not attempt to perform these procedures unless you are qualified to do so.***

Use the procedures in this section to verify that the Keithley Instruments Model 6482's performs within the limits stated in the instrument's one-year accuracy specifications. Perform the verification procedures:

- When you first receive the instrument to make sure that it was not damaged during shipment.
- To verify that the instrument meets factory specifications.
- To determine if calibration is required.
- After performing a calibration adjustment to make sure the instrument was adjusted properly.

NOTE   *If the instrument is still under warranty and its performance is outside specified limits, contact your Keithley Instruments representative or the factory to determine the correct course of action.*

Verification test requirements

Be sure to perform the verification tests:

- Under the proper environmental conditions.
- After the specified warm-up period.
- Using the correct line voltage.
- Using the proper test equipment.
- Using the specified output signals and reading limits.

NOTE   *Product specifications are subject to change. Listed uncertainties and test limits are provided only as examples. Always verify values against actual product specifications.*

Environmental conditions

Conduct performance verification procedures in a test environment with:

- An ambient temperature of 18 °C to 28 °C (65 °F to 82 °F).
- A relative humidity of less than 70 percent, unless otherwise noted.

NOTE   *Product specifications that are listed as 18 ℃ to 28 ℃ assume adjustment has been done at 23 ℃. If the Model 6482 is adjusted at a different temperature, the specifications apply to ±5 ℃ of that adjustment temperature.*

Warm-up period

Allow the Model 6482 to warm up for at least one hour before conducting the verification procedures.

If the instrument has been subjected to temperature extremes (those outside the ranges stated above), allow additional time for the instrument's internal temperature to stabilize. Typically, allow one extra hour to stabilize an instrument that is 10 °C (18 °F) outside the specified temperature range.

Allow the test equipment to warm up for the minimum time specified by the manufacturer.

Line power

The Model 6482 requires a line voltage of 100 V, 120 V, 200 V, or 240 V (depending on rear panel line frequency setting) and a line frequency of 50 Hz or 60 Hz. Verification tests must be performed within this range. Make sure the line voltage setting that is visible through the small window in the rear-panel power module is at the required setting. If not, change the setting, as described in Section 18.

Recommended test equipment

Table 17-1 summarizes recommended verification equipment and pertinent specifications. Total test equipment measurement uncertainty should meet or be less than the listed values at each test point. Generally, test equipment uncertainty should be at least four times better than corresponding Model 6482 specifications.

NOTE   *The sources listed in Table 17-1 do not meet the recommended four times better uncertainty for certain output values. Factory calibration uses a transfer standard technique that yields uncertainties better that 4:1 for all calibration points.*

Table 17-1
Recommended verification equipment

Description	Manufacturer/model	Specifications			Uncertainty ratio[2]
Meter/Source	Keithley Instruments Model 6430 Sub-Femtoamp Remote SourceMeter® instrument with preamp	DC current	1.9 nA: 19 nA: 190 nA: 1.9 µA:	±0.155 % ±0.155 % ±0.208 % ±0.155 %	7.1:1 2.6:1 2:1 1.4:1
Calibrator	Fluke 5700A Multifunction Calibrator	DC current	19 µA: 190 µA: 1.9 mA: 19 mA:	±0.059 % ±0.011 % ±0.007 % ±0.007 %	3.5:1 9.8:1 31:1 17:1
Digital multimeter	Keithley Instruments Model 2001	DC voltage[1]	20 V: 200 V:	±22 ppm ±30 ppm	45:1 33:1

[1] 90-day full-range accuracy specifications.
[2] Ratio between test equipment uncertainty and Model 6482 uncertainty.

Verification limits

The verification limits listed in this section have been calculated using only the Model 6482 one-year accuracy specifications, and they do not include test equipment uncertainty. If a particular measurement falls outside of the allowable range, recalculate new limits based on Model 6482 specifications and corresponding test equipment specifications.

Example limits calculation

As an example of how verification limits are calculated, assume you are testing the 2 mA range with a 1.9 mA input current. Using the Model 6482 one-year accuracy specification of ±(0.1 % of reading + 2 µA offset), the calculated reading limits are:

Limits = 1.9 mA ± [(1.9 mA × 0.1%) + 2 µA]
Limits = 1.9 mA ± (1.9 µA + 2 µA)
Limits = 1.9 mA ± 3.9 µA
Limits = 1.8961 mA to 1.9039 mA

Restoring factory defaults

Before performing the verification procedures, restore the instrument to its factory front-panel bench defaults as follows:

1. Press the **MENU** key. The instrument will display the following prompt:

    ```
    MAIN MENU
    SAVESETUP  COMMUNICATION  CAL  ▶
    ```
2. Select **SAVESETUP** and press the **ENTER** key. The instrument then displays:

    ```
    SAVESETUP MENU
    SAVE  RESTORE  POWERON  RESET
    ```
3. Select **RESET** and press the **ENTER** key. The instrument displays:

    ```
    RESET ORIGINAL DFLTS
    BENCH  GPIB
    ```
4. Select **BENCH** and press the **ENTER** key to restore bench defaults.

Performing the verification test procedures

Test summary

- Current measurement accuracy
- Voltage bias accuracy

If the Model 6482 is not within specifications and not under warranty, see the Adjustment procedures in this section for information on calibrating the instrument.

Test considerations

When performing the verification procedures:

- Be sure to restore factory front-panel defaults, as previously outlined.
- Make sure that the test equipment is fully warmed up and properly connected to the correct Model 6482 terminals as required.
- Allow signals to settle before making a measurement.
- Do not connect test equipment to the Model 6482 through a scanner, multiplexer, or other switching equipment.

WARNING   *The maximum common-mode voltage (voltage between LO and chassis ground) is 200 V DC. Exceeding this value may cause a shock hazard.*

CAUTION   **Maximum signal on OUTPUT connectors is 30 V at 20 mA DC. Maximum signal on INPUT connectors is 20 mA. Exceeding these values may result in damage to the instrument.**

Current measurement accuracy

Follow the steps below to verify that the Model 6482 current measurement accuracy is within specified limits. This test involves applying a current to the Model 6482 and verifying that the measured readings are within required limits. The current applied is from either a Model 6430 or a Model 5700A.

NOTE   *The following steps verify the 2 nA through 2 μA ranges on channel 1.*

1. With the power off, connect the Model 6430 to the channel 1 INPUT triaxial connector through the preamp as shown in Figure 17-1.
2. Turn on the Model 6482 and the Model 6430, and allow them to warm up for at least one hour.
3. Restore front-panel bench defaults as outlined in Restoring factory defaults.
4. On the Model 6482, press the **MSR1** key, and then select the 2 nA range on channel 1 with the RANGE ▼ key.
5. Select the Model 6430 amperes function and choose the 2 nA range.
6. Set the Model 6430 output to 1.90000 nA, and turn the output on.
7. Verify that the Model 6482 current reading is within the limits for the 2 nA range shown in Table 17-2.
8. Repeat steps 4 through 7 for the 20 nA through 2 μA ranges.

NOTE   *The following steps verify the 2 nA through 2 μA ranges on channel 2.*

9. With the power off, connect the Model 6430 to the channel 2 INPUT triaxial connector through the preamp as shown in Figure 17-1.
10. Repeat the procedure for channel 2 (step 4 through step 8). Make sure to select channel 2 by pressing the **MSR2** key.

NOTE   *The following steps verify the 20 μA through 20 mA ranges on channel 1.*

11. With the power off, connect the Model 5700A to the Model 6482 channel 1 INPUT triaxial connector as shown in Figure 17-2.
12. Turn on the Model 6482 and the Model 5700A, and allow them to warm up for at least one hour.
13. On the Model 6482, press the **MSR1** key, and then select the 20 μA range on channel 1 using the RANGE ▲ and ▼ keys (the 20 μA range has the following format: +00.0000 μA).
14. Select the Model 5700A amperes function and choose the 20 μA range.

15. Set the Model 5700A output to 19.0000 µA, and turn the output on.

16. Verify that the Model 6482 current reading is within the limits for the 20 µA range as shown in Table 17-2.

17. Repeat steps 12 through 15 for the 200 µA through 20 mA ranges.

NOTE   *The following steps verify the 20 µA through 20 mA ranges on channel 2.*

18. With the power off, connect the Model 5700A to the Model 6482 channel 2 INPUT triaxial connector as shown in Figure 17-2.

19. Repeat the procedure for channel 2 (step 13 through step 17). Make sure to select channel 2 by pressing the **MSR2** key.

Figure 17-1
Connections for current verification tests (2 nA to 2 µA ranges)
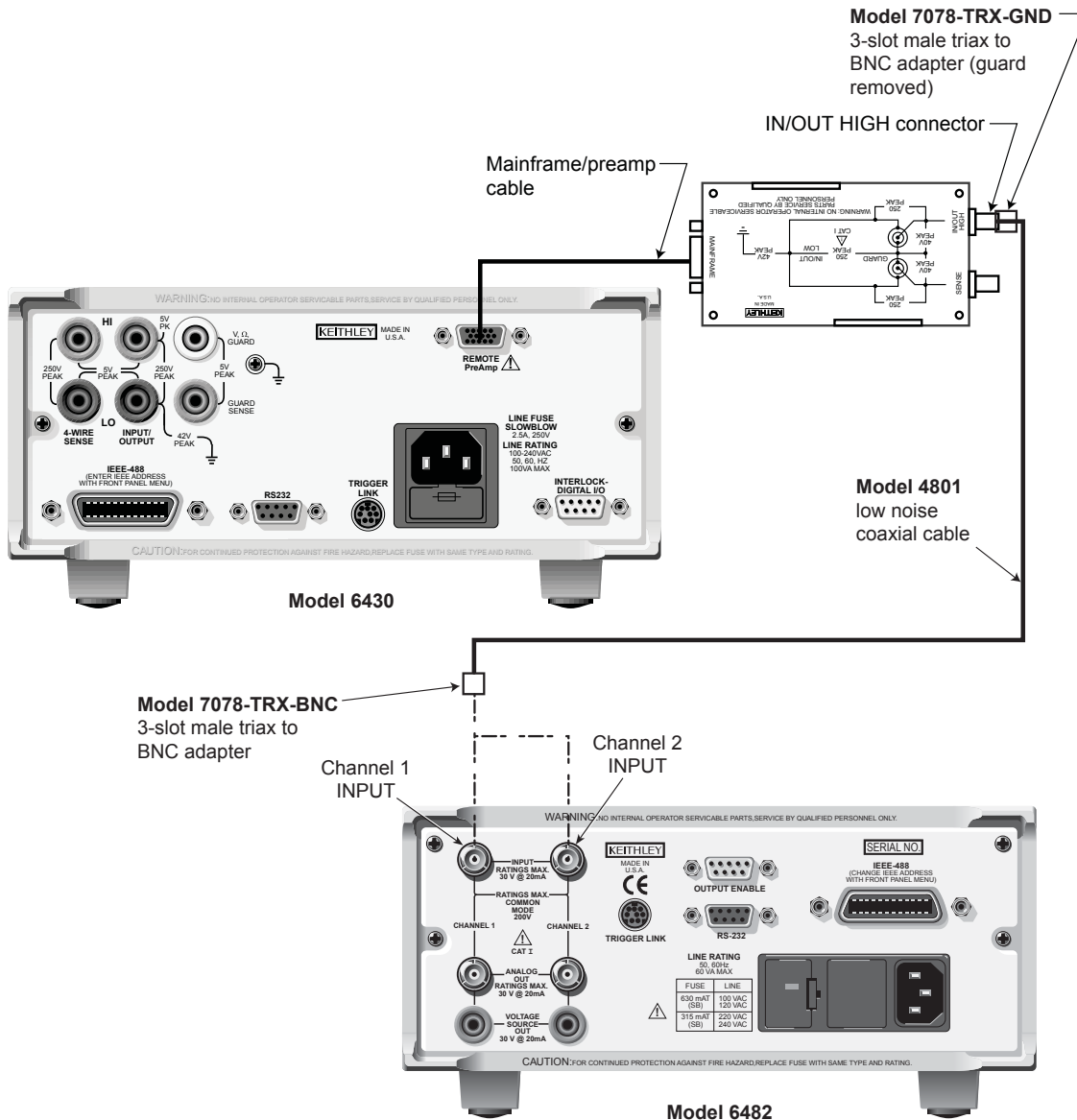
Table 17-2
Current measurement verification limits

Connections	Model 6482 range	Model 6430 current	Model 5700A current	Reading limits* 1 year, 18° C to 28° C (65° F to 82° F)
Figure 17-1	2 nA	1.90000 nA	-	1.87900 nA to 1.92100 nA
	20 nA	19.0000 nA	-	18.9220 nA to 19.0780 nA
	200 nA	190.000 nA	-	189.230 nA to 190.770 nA
	2 μA	1.90000 μA	-	1.89600 μA to 1.90400 μA
Figure 17-2	20 μA	-	19.0000 μA	18.9610 μA to 19.0390 μA
	200 μA	-	190.000 μA	189.790 μA to 190.210 μA
	2 mA	-	1.90000 mA	1.89610 mA to 1.90390 mA
	20 mA	-	19.0000 mA	18.9790 mA to 19.0210 mA

*Limits do not include Model 6430/5700A uncertainty.

Figure 17-2
Connections for current verification tests (20 μA to 20 mA ranges)

Model BG-18
Dual banana to BNC coaxial adapter. Connect output LO to shield.

Model 4801
low noise coaxial cable

Model 7078-TRX-BNC
3-slot male triax to BNC adapter

HI

Output
VΩA

LO

Fluke Model 5700A multifunction calibrator

Channel 1 INPUT

Channel 2 INPUT
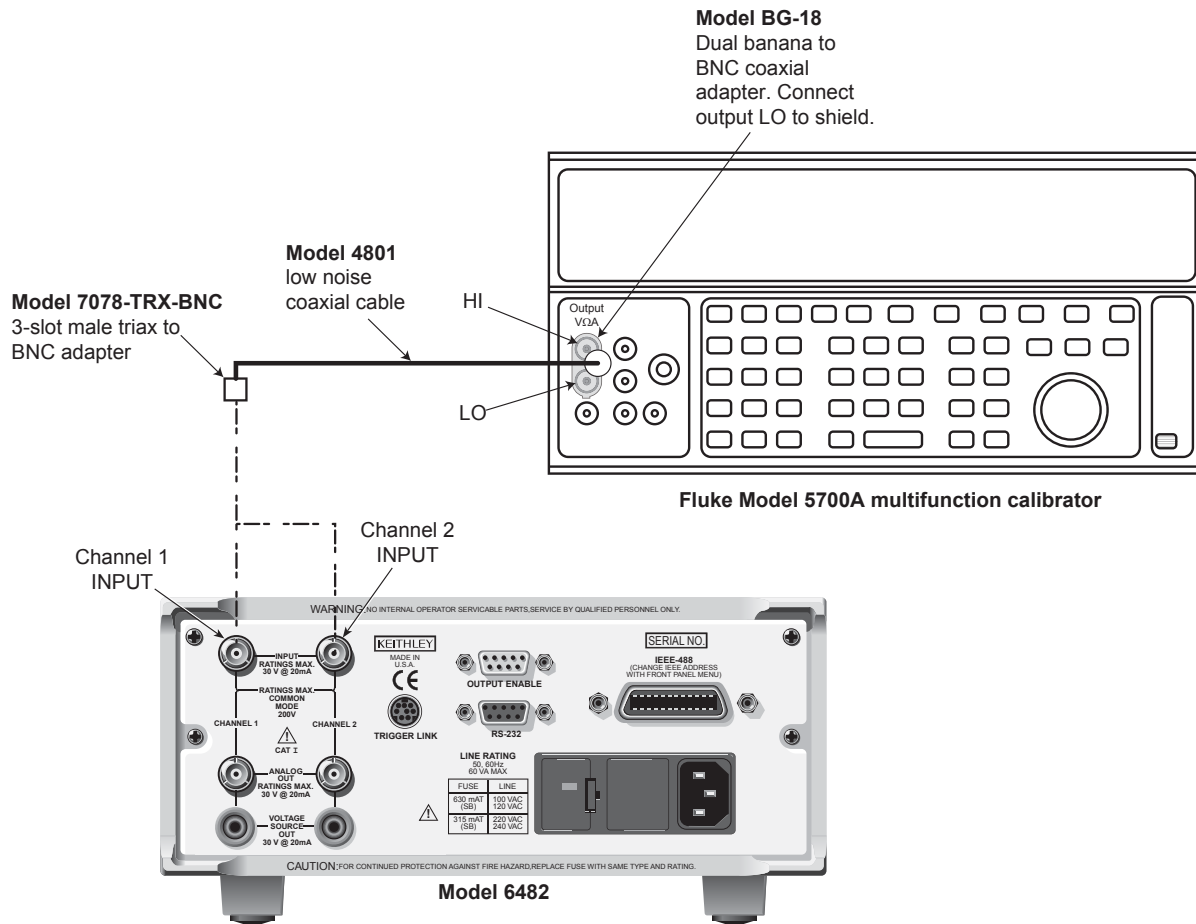
Model 6482

Voltage bias accuracy

Follow the steps below to verify that Model 6482 bias voltage accuracy is within specified limits. This test involves setting the bias voltage to specific values and measuring the voltages with a digital multimeter (DMM).

Figure 17-3
Connections for channel 1 bias voltage verification tests
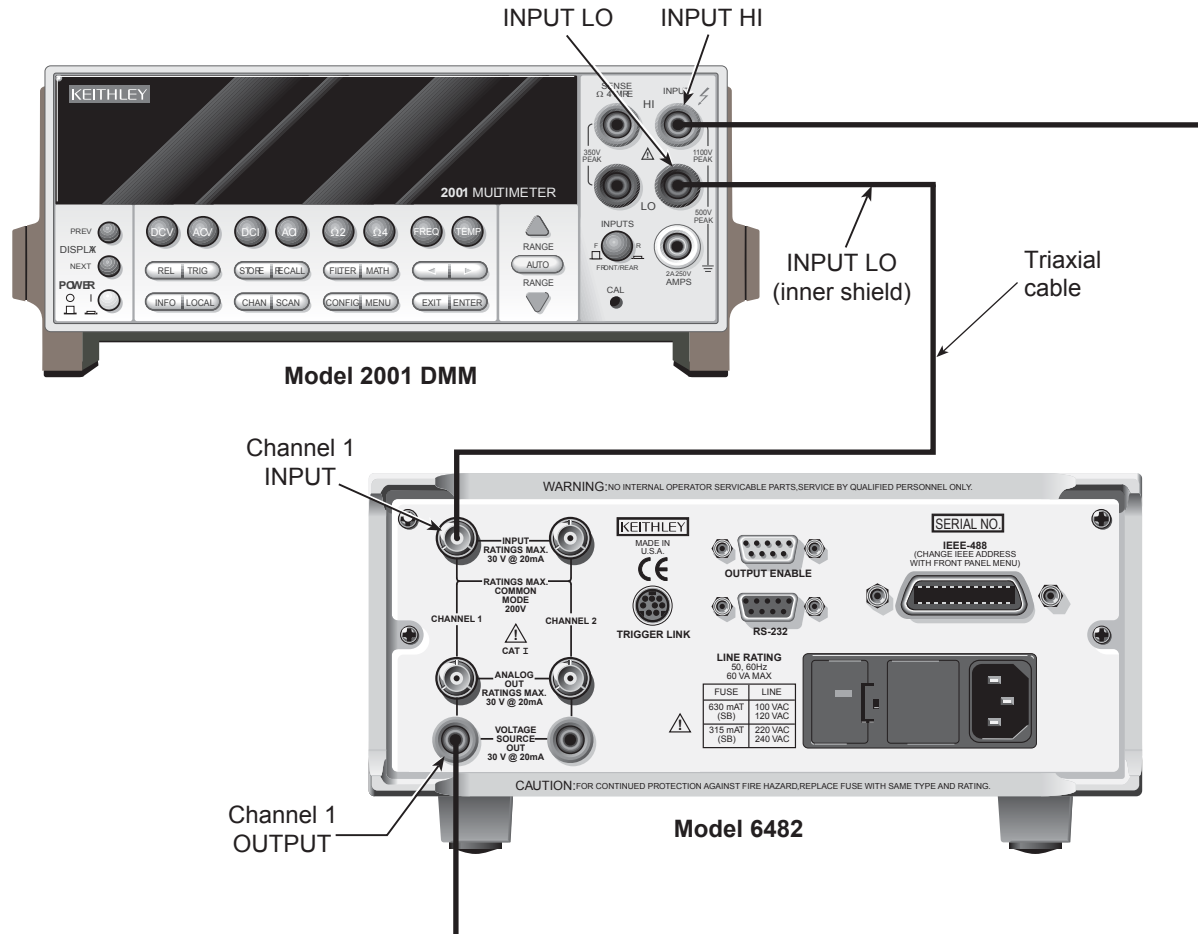
1. With the power off, connect the digital multimeter to the Model 6482 channel 1 INPUT and OUTPUT connectors, as shown in Figure 17-3. Connect the DMM INPUT HI to the VOLTAGE SOURCE OUTPUT CHANNEL 1 connector, and connect the DMM INPUT LO to the inner shield (LO) of the INPUT CHANNEL 1 connector.
2. Turn on the Model 6482 and DMM, and allow them to warm up for at least one hour.
3. Restore bench defaults as outlined in Restoring factory defaults.
4. Select the multimeter DC volts measuring function, and then select autorange.
5. On the Model 6482, press the **SCR1** key, and then select the 10 V bias range on channel 1 with the RANGE ▾ key.
6. Set the channel 1 source voltage to +10.000 V as follows:
   • Press the **SRC1** key. Note that the EDIT indicator turns on, and a digit blinks in the Src1: display field to indicate the instrument is in the edit mode.
   • Set the source value in one of two ways: (1) Enter the value using the numeric keys, or (2) set the value using the EDIT keys. Use EDIT ◄ and ► to place the cursor on the digit to be changed, and use EDIT ▲ and ▾ to increment or decrement the value.
7. Press the **OUTPUT** key to turn on the output.

8. Verify the bias voltage output is within the limits for the 10 V range shown in Table 17-3.
9. Repeat steps 6 and 8 for an output voltage of -10.000 V.
10. Press the **OUTPUT** key to turn off the output.
11. Repeat steps 5 through 10 for the 30 V bias voltage range with both +30.00 V and -30.00 V output voltages.

Table 17-3
Bias voltage accuracy limits

Model 6482 bias range	Bias voltage	Bias voltage limits 1 year, 18° C to 28° C (65° F to 82° F)
10 V	10.000 V	9.98 V to 10.02 V
30 V	30.000 V	29.86 V to 30.14 V

12. After verifying both ranges on channel 1:
    a. With the power off, make connections to the channel 2 INPUT and OUTPUT connectors (see Figure 17-4 for connections)
    b. Repeat steps 5 through 11 verifying the acurracy of channel 2 (press the **SRC2** key to select channel 2).

Figure 17-4
Connections for channel 2 bias voltage verification tests
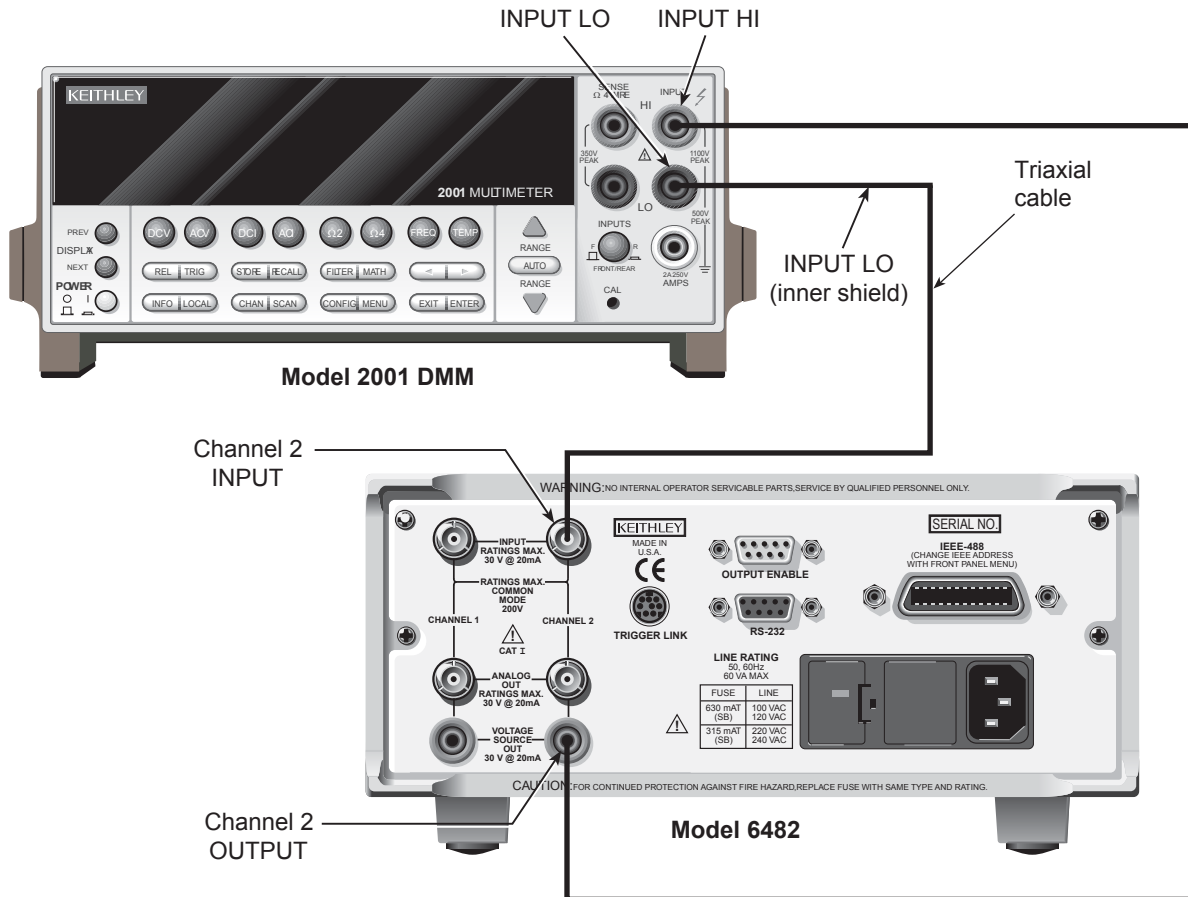
Analog output accuracy

Follow the steps below to verify that Model 6482 analog output voltage accuracy is within specified limits. This test involves applying currents from a Model 6430 and verifying that the 20 mA range analog output voltage readings are within required limits.

> *CAUTION*   **With the ground connect mode enabled, analog output LO can float up to ±30 V above chassis ground, depending on the voltage bias source setting. Connecting the analog outputs to external equipment that does not allow analog output low to float up to ±30 V may result in damage to the Model 6482 voltage bias sources when the ground connect mode is enabled.**

1. With the power off, connect the Model 6430 to the channel 1 INPUT triaxial connector through the preamp (using the Mainframe/preamp cable and a triaxial cable), and connect the DMM to the channel 1 ANALOG OUT connector, as shown in Figure 17-5.
2. Turn on the Model 6482, Model 6430, and the Model 2001 DMM. Allow them to warm up for at least one hour.
3. Restore front-panel bench defaults, as outlined in Restoring factory defaults.
4. On the Model 6482, press the **MSR1** key, and then select the 2 nA range on channel 1 by pressing the RANGE ▾ key as needed.
5. Select the Model 6430 amperes function and choose the 2 nA range.
6. Set the Model 6430 output to 1.90000 nA and turn the output on.
7. Verify that the analog output voltage reading, as measured by the DMM, is within the limits for the 2 nA range shown in Table 17-4.

Table 17-4
Model 6482 analog output verification limits

Model 6482 range	Model 6430 current	Analog output voltage limits* 1 year, 18° C to 28° C (65° F to 82° F)
2 nA	1.90000 nA	-8.840 V to -10.160 V
20 nA	19.0000 nA	-9.206 V to -9.794 V
200 nA	190.000 nA	-8.840 V to -10.160 V
2 µA	1.90000 µA	-9.206 V to -9.794 V
20 µA	19.0000 µA	-8.840 V to -10.160 V
200 µA	190.000 µA	-9.254 V to -9.747 V
2 mA	1.90000 mA	-8.840 V to -10.160 V
20 mA	19.0000 mA	-9.254 V to -9.747 V

*Limits do not include Model 6430 or DMM uncertainty.

8. Repeat steps 4 through 7 for the 20 nA through 20 mA ranges.
9. Repeat steps 4 through 8 for negative input currents of the same magnitude. Analog output voltages will be positive with negative input currents.
10. After verifying all ranges on channel 1:
    a. With the power off, make connections to the channel 2 (see Figure 17-6 for connections).
    b. Repeat steps 3 through 9 verifying the acurracy of channel 2 (press the **MSR2** key to select channel 2).

Figure 17-5
Connections for channel 1 analog output verification
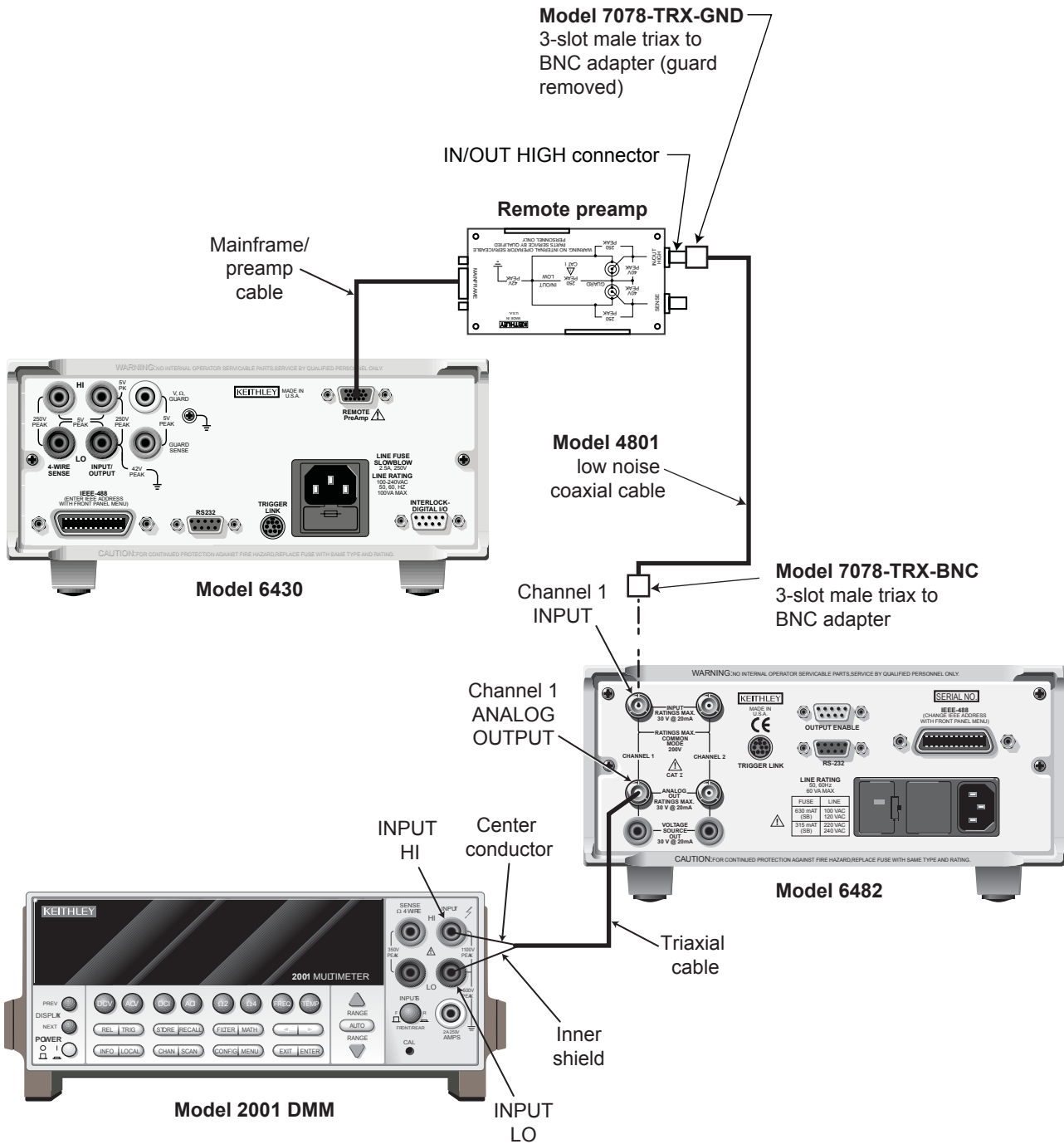
Figure 17-6
Connections for channel 2 analog output verification
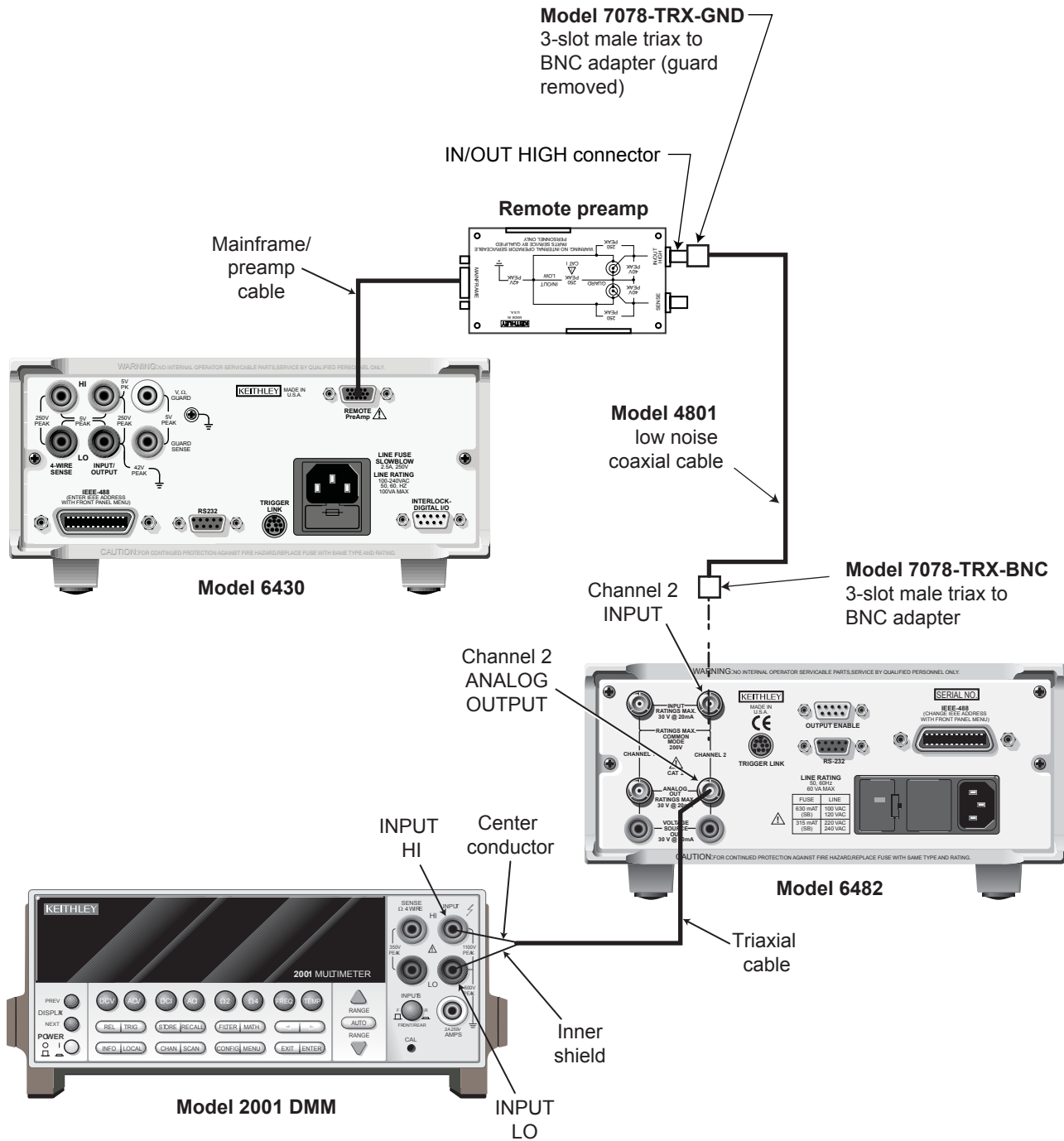
Adjustment

Use the procedures in this section to calibrate the Model 6482. These procedures require accurate test equipment to supply precise currents and measure accurate DC voltages. Calibration can be performed either from the front-panel or by sending SCPI calibration commands over the IEEE-488 bus or RS-232 port, with the aid of a computer.

WARNING **The information in this section is intended for qualified service personnel only. Do not attempt these procedures unless you are qualified to do so. Some of these procedures may expose you to hazardous voltages.**

Environmental conditions

Temperature and relative humidity

Conduct the calibration procedures at an ambient temperature of 18 °C to 28 °C (65 °F to 82 °F), with relative humidity of less than 70 percent (unless otherwise noted).

NOTE *Product specifications that are listed as 18 °C to 28 °C assume adjustment has been done at 23 °C. If the Model 6482 is adjusted at a different temperature, the specifications apply to ±5 °C of that adjustment temperature.*

Warm-up period

Allow the Model 6482 to warm up for at least one hour before conducting the verification procedures.

If the instrument has been subjected to temperature extremes (those outside the ranges stated above), allow additional time for the instrument's internal temperature to stabilize. Typically, allow one extra hour to stabilize an instrument that is 10 °C (18 °F) outside the specified temperature range.

Allow the test equipment to warm up for the minimum time specified by the manufacturer.

Line power

The Model 6482 requires a line voltage of 100 V, 120 V, 200 V, or 240 V (depending on line voltage setting) at a line frequency of 50 Hz or 60 Hz. The instrument must be calibrated while operating from a line voltage within this range. Make sure the line voltage setting, that is visible through the small window in the rear-panel power module, is at the required setting. If not, change the setting, as described in Section 18.

Adjustment considerations

When performing the adjustment procedures:

- Make sure the test equipment is properly warmed up and connected to the Model 6482 input or output terminals, as required.
- Allow signals to settle before calibrating each point.
- Do not connect test equipment to the Model 6482 through a scanner or other switching equipment.
- If an error occurs during calibration, the Model 6482 will generate an error message. See Appendix A for more information.

WARNING   ***The maximum common-mode voltage (voltage between LO and chassis ground) is 200 V DC. Exceeding this value may cause a shock hazard.***

CAUTION   **Maximum signal on OUTPUT connectors is 30 V at 20 mA DC. Maximum signal on INPUT connectors is 20 mA. Exceeding these values may result in damage to the instrument.**

Calibration adjustment cycle

Perform a calibration adjustment at least once a year to ensure the instrument meets or exceeds its specifications.

Recommended calibration adjustment equipment

Table 17-5 lists the recommended equipment for the calibration procedures. You can use alternate equipment if the equipment has specifications equal to or greater than those listed in the table. When possible, test equipment specifications should be at least four times better than corresponding Model 6482 specifications.

NOTE   *The sources listed in Table 17-5 do not meet the recommended four times better uncertainty for certain output values. The Model 6482 may not meet factory calibration accuracy specifications on these ranges, unless further steps are taken to characterize the corresponding source ranges using more accurate equipment. Factory calibration uses a transfer standard technique that yields uncertainties better than 4:1 for all calibration points.*

Table 17-5
Recommended calibration equipment

Description	Manufacturer/model	Specifications			Uncertainty ratio[2]
Meter/Source	Keithley Instruments Model 6430 Sub-Femtoamp Remote SourceMeter® instrument with preamp	DC current	1.9 nA: 19 nA: 190 nA: 1.9 μA:	±0.155 % ±0.155 % ±0.208 % ±0.155 %	7.1:1 2.6:1 2:1 1.4:1
Calibrator	Fluke 5700A Multifunction Calibrator	DC current	19 μA: 190 μA: 1.9 mA: 19 mA:	±0.059 % ±0.011 % ±0.007 % ±0.007 %	3.5:1 9.8:1 31:1 17:1
Digital multimeter	Keithley Instruments Model 2001	DC voltage[1]	20 V: 200 V:	±22 ppm ±30 ppm	45:1 33:1

[1] 90-day full-range accuracy specifications.
[2] Ratio between test equipment uncertainty and Model 6482 uncertainty.

Calibration menu

Table 17-6 summarizes the main calibration menu selections. To enter the calibration menu, press the **MENU** key, select **CAL**, then press the **ENTER** key. Use the EDIT keys to move the cursor and scroll through menu selections. Press the **ENTER** key to select a menu item.

Table 17-6
Calibration menu

Menu selection	Description
UNLOCK	Unlock calibration using password (default: 006482).
EXECUTE	Execute calibration steps.
V-CAL	Voltage bias calibration.
CHAN-1	Channel 1 voltage bias calibration.
CHAN-2	Channel 2 voltage bias calibration.
I-CAL	Current measurement calibration.
CHAN-1	Channel 1 current calibration.
CHAN-2	Channel 2 current calibration.
OFFSET	Autozero input offset for both channels.
VIEW-DATES	View calibration dates.
SAVE	Save calibration constants.
LOCK	Lock out calibration.
CHANGE-PASSWORD	Change calibration password.

Unlocking calibration

Before performing calibration, you must first unlock calibration by entering or sending the calibration password, as explained in the following paragraphs.

Unlocking calibration from the front-panel

1. Press the **MENU** key, select **CAL**, and then press the **ENTER** key. The instrument will display the following:

   ```
   CALIBRATION
   UNLOCK EXECUTE VIEW-DATES  ▶
    ◀  SAVE LOCK CHANGE-PASSWORD
   ```

2. Select **UNLOCK**, and then press the **ENTER** key. The instrument will display the following:

   ```
   PASSWORD:
   Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
   ```

3. Use the EDIT ▲ and ▼ keys to select the letter or number, and use the EDIT ◀ and ▶ arrow keys to choose the position. Press ▼ for letters; ▲ for numbers. Enter the present password on the display. Front-panel default: 006482.

4. When the correct password is displayed, press the **ENTER** key. You can then proceed with the calibration procedure.

Unlocking calibration through a remote interface

To unlock calibration over a remote interface, send the following command:

   ```
   :CAL:PROT:CODE   '<password>'
   ```

For example, the following command uses the default password:

   ```
   :CAL:PROT:CODE   'KI006482'
   ```

Changing the password

The default password (KI006482) may be changed from the front-panel or over a remote interface, as discussed below.

Changing the password from the front-panel

Follow these steps to change the password from the front-panel:

1. Press the **MENU** key, select **CAL**, and then press the **ENTER** key. The instrument will display the following:

   ```
   CALIBRATION
   UNLOCK EXECUTE VIEW-DATES  ▶
      ◀  SAVE LOCK CHANGE-PASSWORD
   ```

2. Select **UNLOCK** then enter the password. Default: 006482.

3. Select **CHANGE-PASSWORD** and press the **ENTER** key. The instrument will display the following:

   ```
   New Pwd: 006482
   Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
   ```

4. Using the EDIT keys, enter the new password on the display.

5. When finished entering the new password, press the **ENTER** key to store the new password.

Changing the password through a remote interface

To change the calibration password through a remote interface, first send the present password, and then send the new password. For example, the following command sequence changes the password from the 'KI006482' remote default to 'KI_CAL':

   ```
   :CAL:PROT:CODE   'KI006482'

   :CAL:PROT:CODE   'KI_CAL'
   ```

You can use any combination of letters and numbers up to a maximum of eight characters.

NOTE   *If you change the first two characters of the password to something other than "KI", you will not be able to unlock calibration from the front-panel.*

Resetting the calibration password

If you lose the calibration password, you can unlock calibration by shorting together the CAL pads, which are located on the display board. Doing so will also reset the password to the factory default (006482, front-panel; KI006482, remote).

Viewing calibration dates and calibration count

When calibration is locked, only the UNLOCK and VIEW-DATES selections will be accessible in the calibration menu. To view calibration dates and calibration count at any time:

1. From normal display, press the **MENU** key, select **CAL**, and then press the **ENTER** key. The instrument will display the following:

    ```
    CALIBRATION
    UNLOCK EXECUTE VIEW-DATES  ▶
    ```

2. Select VIEW-DATES and then press the **ENTER** key. The Model 6482 will display the next and last calibration dates and the calibration count, as shown in the following example:

    ```
    NEXT CAL: 07/31/2013
    Last cal: 07/31/2012  Count: 0001
    ```

Calibration adjustment errors

The Model 6482 checks for errors after each calibration step, minimizing the possibility that improper calibration may occur due to operator error.

Front-panel error reporting

If an error is detected during comprehensive calibration, the instrument will display an appropriate error message (see Appendix A). The instrument will then prompt you to repeat the calibration step that caused the error.

Remote error reporting

You can detect errors from a remote interface by testing the state of EAV (Error Available) bit (bit 2) in the status byte. Use the *STB? query to request the status byte. Query the instrument for the type of error by using the :SYST:ERR? query. The Model 6482 will respond with the error number and a text message describing the error. See Appendix A for details.

Aborting calibration steps

To abort a calibration step from the front-panel, press the **EXIT** key. To abort a calibration step over a remote interface, send the :ABORt command.

Front-panel calibration

The front-panel calibration procedure described below calibrates all functions. Note that each function and range is separately calibrated, and the procedure must be performed in the order shown.

Step 1. Prepare the Model 6482 for calibration adjustment

1. Turn on the Model 6482 and the calibration equipment, and allow them to warm up for at least one hour before performing calibration.
2. Press the **MENU** key, select **CAL**, and then press the **ENTER** key. Select **UNLOCK**, and then press the **ENTER** key. The instrument will display the following:

    ```
    PASSWORD:
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

3. Use the EDIT ▲ and ▼ keys to select the letter or number, and use the ◀ and ▶ arrow keys to choose the position. Press EDIT ▼ for letters; ▲ for numbers. Enter the present password on the display. Front-panel default: 006482.

4. Press the **ENTER** key to complete the process.

5. Press the **EXIT** key to return to normal display.

Table 17-7
Front-panel calibration summary

Function[1]	Calibration step[2]	Test connections
OFFSET	INPUT offset voltage calibration	Channel 1 and channel 2 INPUT connectors left open (capped).
V-CAL CHAN-1	Voltage bias calibration Channel 1 calibration Positive full-scale output Zero output Negative full-scale output	DMM to channel 1 OUTPUT and INPUT connectors (Figure 17-7) DMM to channel 1 OUTPUT and INPUT connectors (Figure 17-7) DMM to channel 1 OUTPUT and INPUT connectors (Figure 17-7)
CHAN-2	Channel 2 calibration Positive full-scale output Zero output Negative full-scale output	DMM to channel 2 OUTPUT and INPUT connectors (Figure 17-8) DMM to channel 2 OUTPUT and INPUT connectors (Figure 17-8) DMM to channel 2 OUTPUT and INPUT connectors (Figure 17-8)
I-CAL CHAN-1	Current calibration Channel 1 calibration Positive full-scale input Zero input Negative full-scale input	2 nA to 2 µA ranges (Model 6430) Source to channel 1 INPUT connector (Figure 17-9) Source to channel 1 INPUT connector (Figure 17-9) Source to channel 1 INPUT connector (Figure 17-9) 20 µA to 20 mA ranges (Model 5700A)
	Positive full-scale input Zero input Negative full-scale input	Source to channel 1 INPUT connector (Figure 17-10) Source to channel 1 INPUT connector (Figure 17-10) Source to channel 1 INPUT connector (Figure 17-10)
CHAN-2	Channel 2 calibration Positive full-scale input Zero input Negative full-scale input	2 nA to 2 µA ranges (Model 6430) Source to channel 2 INPUT connector (Figure 17-11) Source to channel 2 INPUT connector (Figure 17-11) Source to channel 2 INPUT connector (Figure 17-11) 20 µA to 20 mA ranges (Model 5700A)
	Positive full-scale input Zero input Negative full-scale input	Source to channel 2 INPUT connector (Figure 17-12) Source to channel 2 INPUT connector (Figure 17-12) Source to channel 2 INPUT connector (Figure 17-12)

[1] CAL EXECUTION menu selections.
[2] Steps repeated separately for each range.

Step 2. Input offset voltage calibration adjustment

1. Install a triaxial shielding cap on both INPUT connectors.

2. Select **OFFSET** from the CAL EXECUTION menu, and then press the **ENTER** key. The instrument will display:

```
CURRENT OFFSET CAL
Input 0 A then press ENTER
```

3. Press the **ENTER** key to complete input voltage calibration.

NOTE   *This step calibrates offset voltage for both channels.*

Step 3. Channel 1 voltage source calibration adjustment

Follow the steps below to calibrate both channel 1 bias voltage ranges. Table 17-8 summarizes calibration ranges and voltages.

1. Connect the DMM to the Model 6482 channel 1 INPUT and OUTPUT connectors, as shown in Figure 17-7. Connect the DMM INPUT HI to the VOLTAGE SOURCE OUTPUT CHANNEL 1 connector; connect DMM INPUT LO to the inner shield of the INPUT CHANNEL 1 connector.

2. Select the DMM DC volts function, and then enable autorange.

3. From the normal display, press the Model 6482 **SRC1** key, then set the channel 1 voltage bias source to the 10 V range using the RANGE ▾ key.

4. Press the **MENU** key, select **CAL**, and then press the **ENTER** key.

5. Select **EXECUTE**, and then press the **ENTER** key to enter the CAL EXECUTION menu.

6. Select **V-CAL** and then press the **ENTER** key. The instrument will display the following:

    ```
    V-CAL CHANNEL SELECT
    CHAN-1  CHAN-2
    ```

7. Select **CHAN-1** and press the **ENTER** key. The instrument will prompt for +10 V calibration:

    ```
    V-CAL[1]
    Press ENTER to Output +10.000 V
    ```

8. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: 10.0000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

9. Note the DMM voltage reading, then use the EDIT keys to adjust the Model 6482 display value to match the noted reading.

10. Press the **ENTER** key. The instrument will prompt for the 0 V calibration point:

    ```
    V-CAL[1]
    Press ENTER to Output 00.000 V
    ```

11. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: 00.00000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

12. Note the DMM voltage reading, and then use the EDIT keys to adjust the Model 6482 display value to match the noted reading.

13. Press the **ENTER** key. The instrument will display the prompt for the -10 V calibration point:

    ```
    V-CAL[1]
    Press ENTER to Output -10.000 V
    ```

14. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: -10.00000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

15. Note the DMM voltage reading, then use the EDIT keys to adjust the Model 6482 display value to match the noted reading and press the **ENTER** key.

16. Press the **EXIT** key to return to normal display.

17. Using the appropriate RANGE key select the 30 V bias voltage range.

18. Repeat steps 4 through 15 for the 30 V bias voltage range.

Table 17-8
Voltage bias calibration values

Voltage bias range	Calibration voltages
10 V	+10.0000 V 0.0000 V -10.0000 V
30 V	+30.000 V 0.000 V -30.000 V

Figure 17-7
Channel 1 bias voltage calibration connections
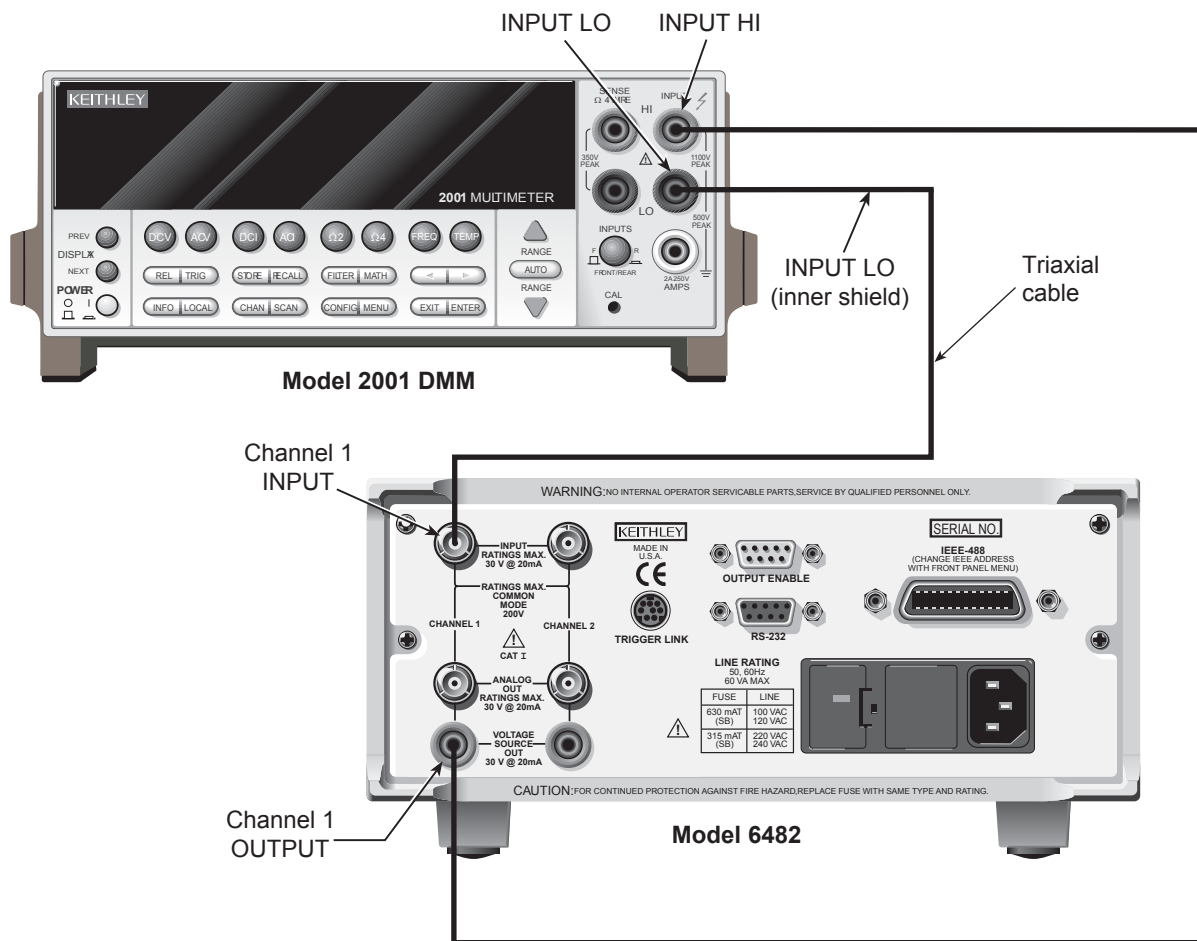
Step 4. Channel 2 voltage source calibration adjustment

Follow the steps below to calibrate both channel 2 bias voltage ranges. Table 17-8 summarizes calibration ranges and voltages.

1. Connect the DMM to the Model 6482 channel 2 INPUT and OUTPUT connectors, as shown in Figure 17-8. Connect the DMM INPUT HI to the VOLTAGE SOURCE OUTPUT CHANNEL 2 connector; connect the DMM INPUT LO to the inner shield of the INPUT CHANNEL 2 connector.

2. Select the DMM DC volts function, and enable autorange.

3. From normal display, press the Model 6482 **SRC2** key, and then set the channel 2 voltage bias source to the 10 V range using the RANGE ▾ key.

4. Press the **MENU** key, select **CAL** and press the **ENTER** key.

5. Select **EXECUTE** and press the **ENTER** key to enter the CAL EXECUTION menu.

6. Select **V-CAL** and press the **ENTER** key. The instrument will display the following:

    ```
    V-CAL CHANNEL SELECT
    CHAN-1   CHAN-2
    ```

7. Select **CHAN-2** and press the **ENTER** key. The instrument will prompt for +10 V calibration:

    ```
    V-CAL²
    Press ENTER to Output +10.000 V
    ```

8. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: 10.0000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

9. Note the DMM voltage reading, and then use the EDIT keys to adjust the Model 6482 display value to match the noted reading.

10. Press the **ENTER** key. The instrument will prompt for the 0 V calibration point:

    ```
    V-CAL²
    Press ENTER to Output 00.000 V
    ```

11. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: 00.00000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

12. Note the DMM voltage reading, and then use the EDIT keys to adjust the Model 6482 display value to match the noted reading.

13. Press the **ENTER** key. The instrument will display the prompt for the -10 V calibration point:

    ```
    V-CAL²
    Press ENTER to Output -10.000 V
    ```

14. Press the **ENTER** key. The instrument will display the following message:

    ```
    DMM RDG: -10.00000 V
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

15. Note the DMM voltage reading, and then use the EDIT keys to adjust the Model 6482 display value to match the noted reading and press the **ENTER** key.

16. Press the **EXIT** key to return to normal display.

17. Using the appropriate RANGE key, select the 30 V bias voltage range.

18. Repeat steps 4 through 15 for the 30 V bias voltage range.

Figure 17-8
Channel 2 bias voltage calibration connections
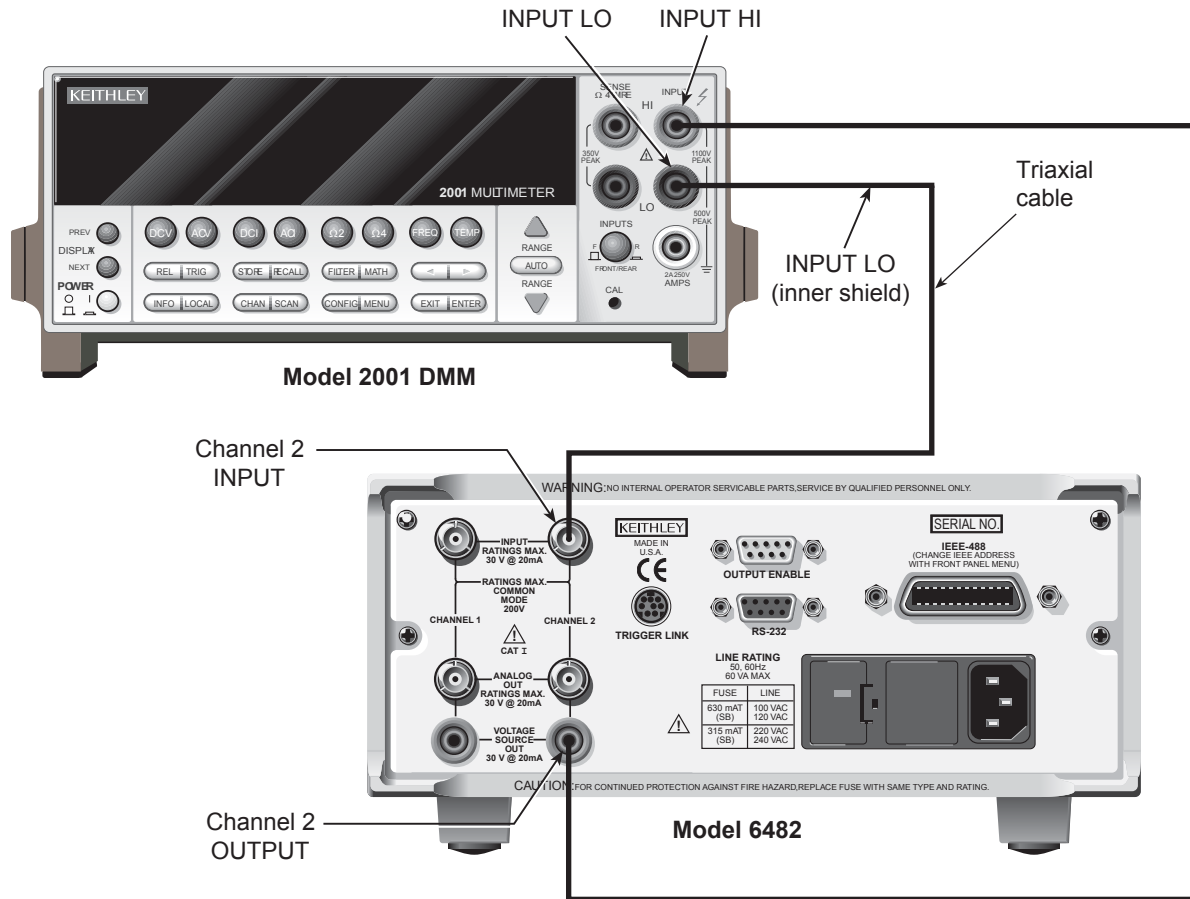
Step 5. Channel 1 current calibration adjustment

Follow the steps below to calibrate all current ranges of channel 1. Table 17-9 summarizes calibration ranges and currents.

1. Connect the Model 6430 to the Model 6482 channel 1 INPUT connector, as shown in Figure 17-9.
2. Select the Model 6430 V/R amperes function, set the Model 6430 output to +1.90000 nA, and then turn on the output.
3. Press the Model 6482 **MSR1** key, and then set channel 1 to the 2 nA current range using the RANGE ▼ key.
4. Press the **MENU** key, and then select **CAL** and press the **ENTER** key.
5. Select **EXECUTE**, and then press the **ENTER** key to enter the CAL EXECUTION menu.
6. Select **I-CAL** and press the **ENTER** key. The instrument will display the following:

    ```
    I-CAL CHANNEL SELECT
    CHAN-1  CHAN-2
    ```

7. Select **CHAN-1** and press the **ENTER** key. The instrument will prompt for the positive full-range current:

    ```
    I (+FS)[1] = +2.000000 nA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

8. Set the Model 6430 current to +1.90000 nA.
9. Use the EDIT keys to adjust the Model 6482 display value to match the Model 6430 current.

10. Press the **ENTER** key. The instrument will prompt for the zero current:
    ```
    I (0)¹ = +0.000000 nA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

11. Set the Model 6430 current to 0 nA and press the **ENTER** key. The instrument will prompt for the negative full-range current:
    ```
    I(-FS)¹ = -2.000000 nA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

12. Set the Model 6430 current to -1.90000 nA.

13. Adjust the displayed current to match the Model 6430 current, and then press the **ENTER** key to complete calibration of the present range.

14. Press the **EXIT** key to return to normal display.

15. Repeat steps 3 through 14 for the 20 nA to 2 µA ranges, using Table 17-9 as a guide. Be sure to set the Model 6482 to the correct range using the RANGE ▲ and ▼ keys, and use the corresponding Model 6430 currents for each range.

16. Connect the Model 5700A to the Model 6482 channel 1 INPUT connector, as shown in Figure 17-10.

Table 17-9
Current calibration values

Connections	Current range	Model 6430 calibration currents	Model 5700A calibration currents
Figure 17-9	2 nA	+1.90000 nA 0.0000 nA -1.90000 nA	- - -
	20 nA	+19.0000 nA 0.0000 nA -19.0000 nA	- - -
	200 nA	+190.000 nA 0.000 nA -190.000 nA	- - -
	2 µA	+1.90000 µA 0.00000 µA -1.90000 µA	- - -
Figure 17-10	20 µA	- - -	+19.0000 µA 0.0000 µA -19.0000 µA
	200 µA	- - -	+190.000 µA 0.000 µA -190.000 µA
	2 mA	- - -	+1.90000 mA 0.00000 mA -1.90000 mA
	20 mA	- - -	+19.0000 mA 0.0000 mA -19.0000 mA

17. Select the Model 5700A V/R amperes function and set the Model 5700A output to +19.0000 µA, and then turn on the output.

18. Press the Model 6482 **MSR1** key, and then set channel 1 to the 20 µA current range using the RANGE ▲ and ▼ keys (the 20 µA range has the following format: +00.0000 µA).

19. Press the **MENU** key, select **CAL**, and then press the **ENTER** key.

20. Select **EXECUTE**, and then press the **ENTER** key to enter the CAL EXECUTION menu.

21. Select **I-CAL** and press the **ENTER** key. The instrument will display the following:

    ```
    I-CAL CHANNEL SELECT
    CHAN-1  CHAN-2
    ```

22. Select **CHAN-1** and press the **ENTER** key. The instrument will prompt for the positive full-range current:

    ```
    I (+FS)¹ = +20.00000 uA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
    ```

    $I (+FS)^1 = +20.00000$ uA

23. Set the Model 5700A current to +19.0000 µA.

24. Use the EDIT keys to adjust the Model 6482 display value to match the Model 5700A current.

25. Press the **ENTER** key. The instrument will prompt for the zero current:

    $I (0)^1 = +00.00000$ uA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.

26. Set the Model 5700A current to 0 nA and press the **ENTER** key. The instrument will prompt for the negative full-range current:

    $I(-FS)^1 = -20.00000$ uA
    Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.

27. Set the Model 5700A current to -19.0000 µA.

28. Adjust the displayed current to match the Model 5700A current, and then press the **ENTER** key to complete calibration of the present range.

29. Press the **EXIT** key to return to normal display.

30. Repeat steps 18 through 29 for the 200 µA to 20 mA ranges, using Table 17-9 as a guide. Be sure to set the Model 6482 to the correct range using the RANGE ▲ and ▼ keys, and use the corresponding Model 5700A currents for each range.

Figure 17-9

Channel 1 current calibration connections (2 nA to 2 μA ranges)
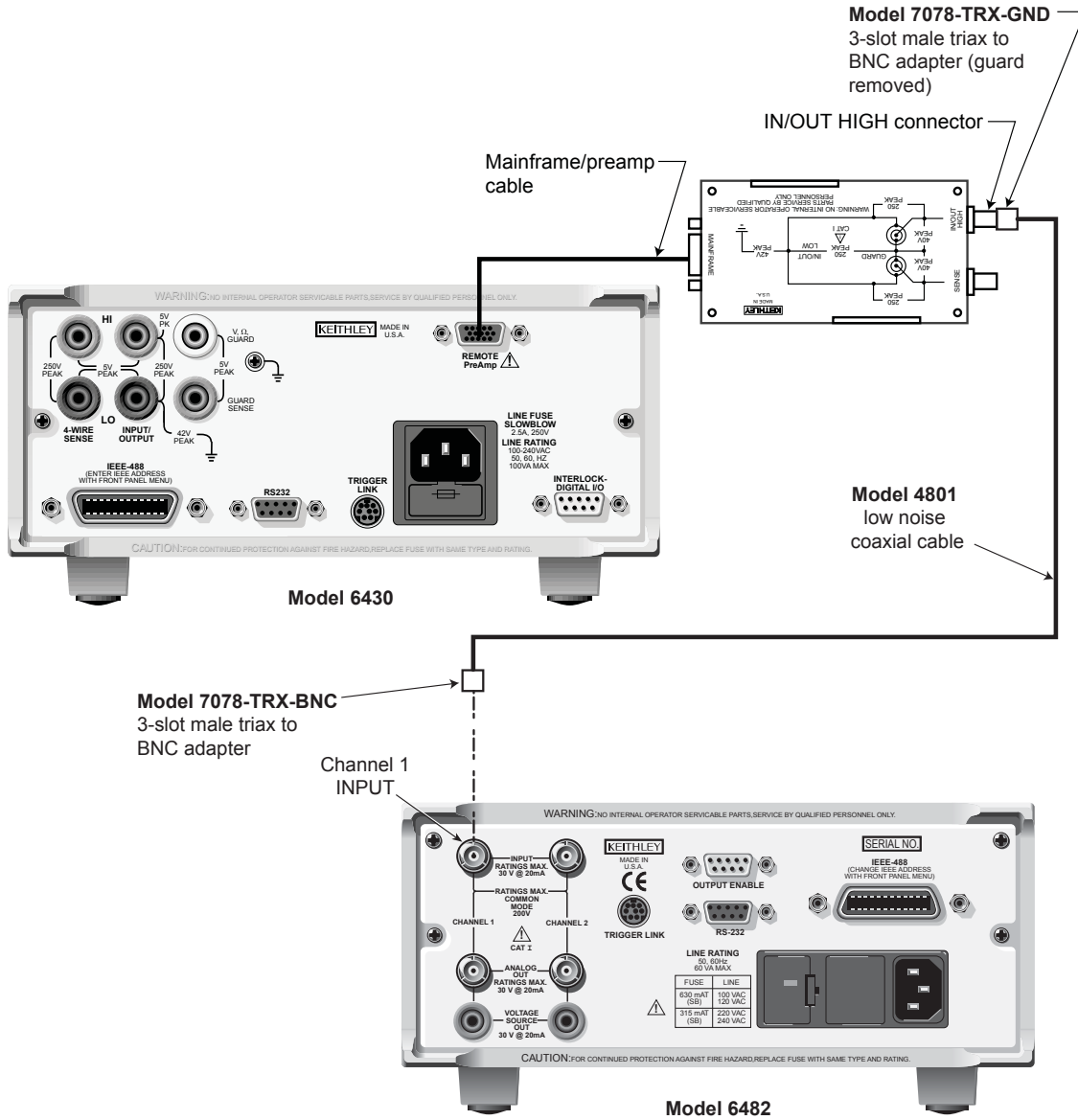
Figure 17-10
Channel 1 current calibration connections (20 μA to 20 mA)
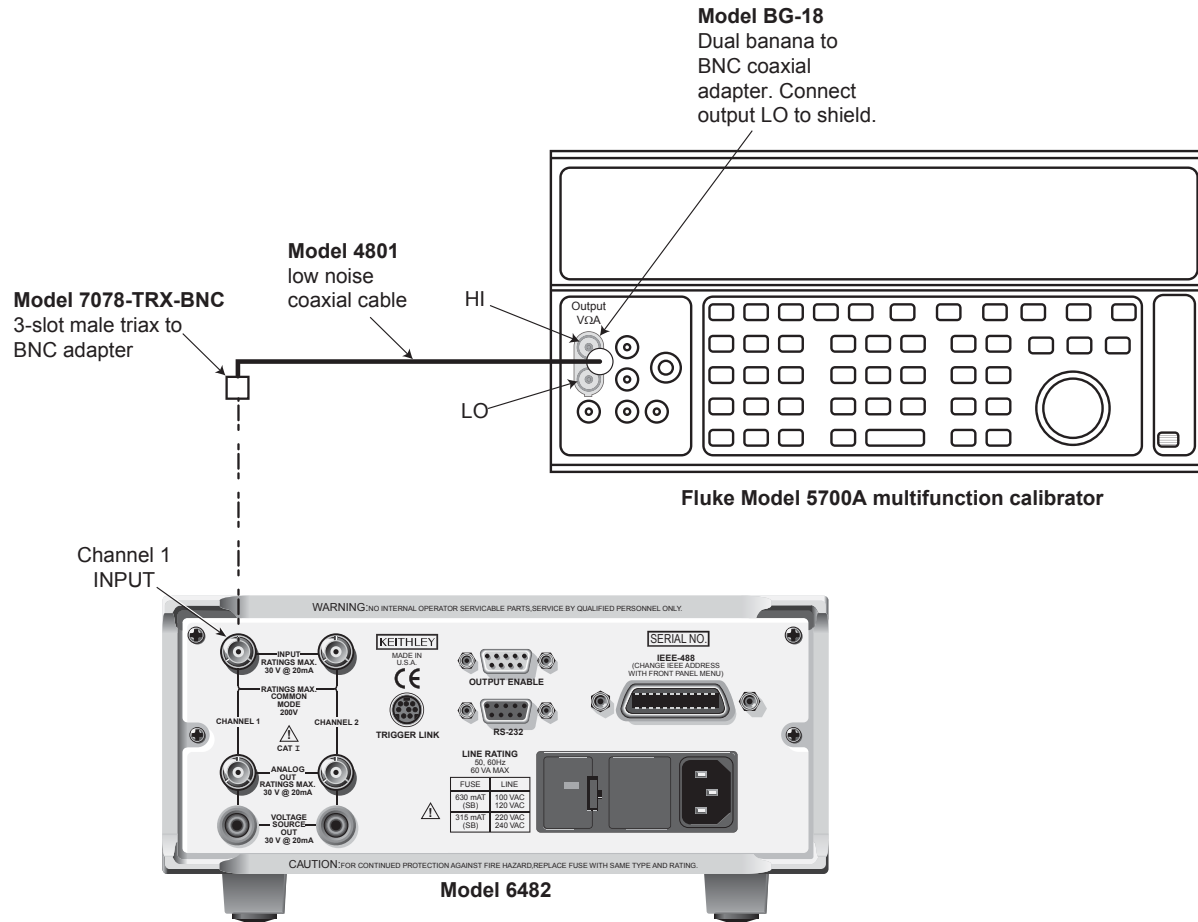
Step 6. Channel 2 current calibration adjustment

Follow the steps below to calibrate all current ranges of channel 2. Table 17-9 summarizes calibration ranges and currents.

1. Connect the Model 6430 to the Model 6482 INPUT CHANNEL 2 connector, as shown in Figure 17-11.
2. Select the Model 6430 V/R amperes function, set the Model 6430 output to +1.90000 nA, and then turn on the output.
3. Press the Model 6482 **MSR2** key, and then set channel 1 to the 2 nA current range using the RANGE ▼ key.
4. Press the **MENU** key, and then select **CAL** and press the **ENTER** key.
5. Select **EXECUTE**, and then press the **ENTER** key to enter the CAL EXECUTION menu.
6. Select **I-CAL** and press the **ENTER** key. The instrument will display the following:

   ```
   I-CAL CHANNEL SELECT
   CHAN-1   CHAN-2
   ```
7. Select **CHAN-2** and press the **ENTER** key. The instrument will prompt for the positive full-range current:

   ```
   I (+FS)² = +2.000000 nA
   Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
   ```
8. Set the Model 6430 current to +1.90000 nA.
9. Use the EDIT keys to adjust the Model 6482 display value to match the Model 6430 current.

10. Press the **ENTER** key. The instrument will prompt for the zero current:

    ```
    I (0)² = +0.000000 nA
    Use ◄ , ► , ▲ , ▼ , ENTER or EXIT.
    ```

11. Set the Model 6430 current to 0 nA and press the **ENTER** key. The instrument will prompt for the negative full-range current:

    ```
    I(-FS)² = -2.000000 nA
    Use ◄ , ► , ▲ , ▼ , ENTER or EXIT.
    ```

12. Set the Model 6430 current to -1.90000 nA.

13. Adjust the displayed current to match the Model 6430 current, and then press the **ENTER** key to complete calibration of the present range.

14. Press the **EXIT** key to return to normal display.

15. Repeat steps 3 through 14 for the 20 nA to 2 μA ranges, using Table 17-9 as a guide. Be sure to set the Model 6482 to the correct range using the RANGE ▲ and ▼ keys, and use the corresponding Model 6430 currents for each range.

16. Connect the Model 5700A to the Model 6482 channel 1 INPUT connector, as shown in Figure 17-12.

17. Select the Model 5700A V/R amperes function and set the Model 5700A output to +19.0000 μA, and then turn on the output.

18. Press the Model 6482 **MSR2** key, and then set channel 1 to the 20 μA current range using the RANGE ▲ and ▼ keys (the 20 μA range has the following format: +00.0000 μA).

19. Press the **MENU** key, select **CAL**, and then press the **ENTER** key.

20. Select **EXECUTE**, and then press the **ENTER** key to enter the CAL EXECUTION menu.

21. Select **I-CAL** and press the **ENTER** key. The instrument will display the following:

    ```
    I-CAL CHANNEL SELECT
    CHAN-1  CHAN-2
    ```

22. Select **CHAN-2** and press the **ENTER** key. The instrument will prompt for the positive full-range current:

    ```
    I (+FS)² = +20.00000 uA
    Use ◄ , ► , ▲ , ▼ , ENTER or EXIT.
    ```

23. Set the Model 5700A current to +19.0000 μA.

24. Use the EDIT keys to adjust the Model 6482 display value to match the Model 5700A current.

25. Press the **ENTER** key. The instrument will prompt for the zero current:

    ```
    I (0)² = +00.00000 uA
    Use ◄ , ► , ▲ , ▼ , ENTER or EXIT.
    ```

26. Set the Model 5700A current to 0 nA and press the **ENTER** key. The instrument will prompt for the negative full-range current:

    ```
    I(-FS)² = -20.00000 uA
    Use ◄ , ► , ▲ , ▼ , ENTER or EXIT.
    ```

27. Set the Model 5700A current to -19.0000 μA.

28. Adjust the displayed current to match the Model 5700A current, and then press the **ENTER** key to complete calibration of the present range.

29. Press the **EXIT** key to return to normal display.

30. Repeat steps 18 through 29 for the 200 μA to 20 mA ranges, using Table 17-9 as a guide. Be sure to set the Model 6482 to the correct range using the RANGE ▲ and ▼ keys, and use the corresponding Model 5700A currents for each range.

Figure 17-11
Channel 2 current calibration connections (2 nA to 2 μA ranges)
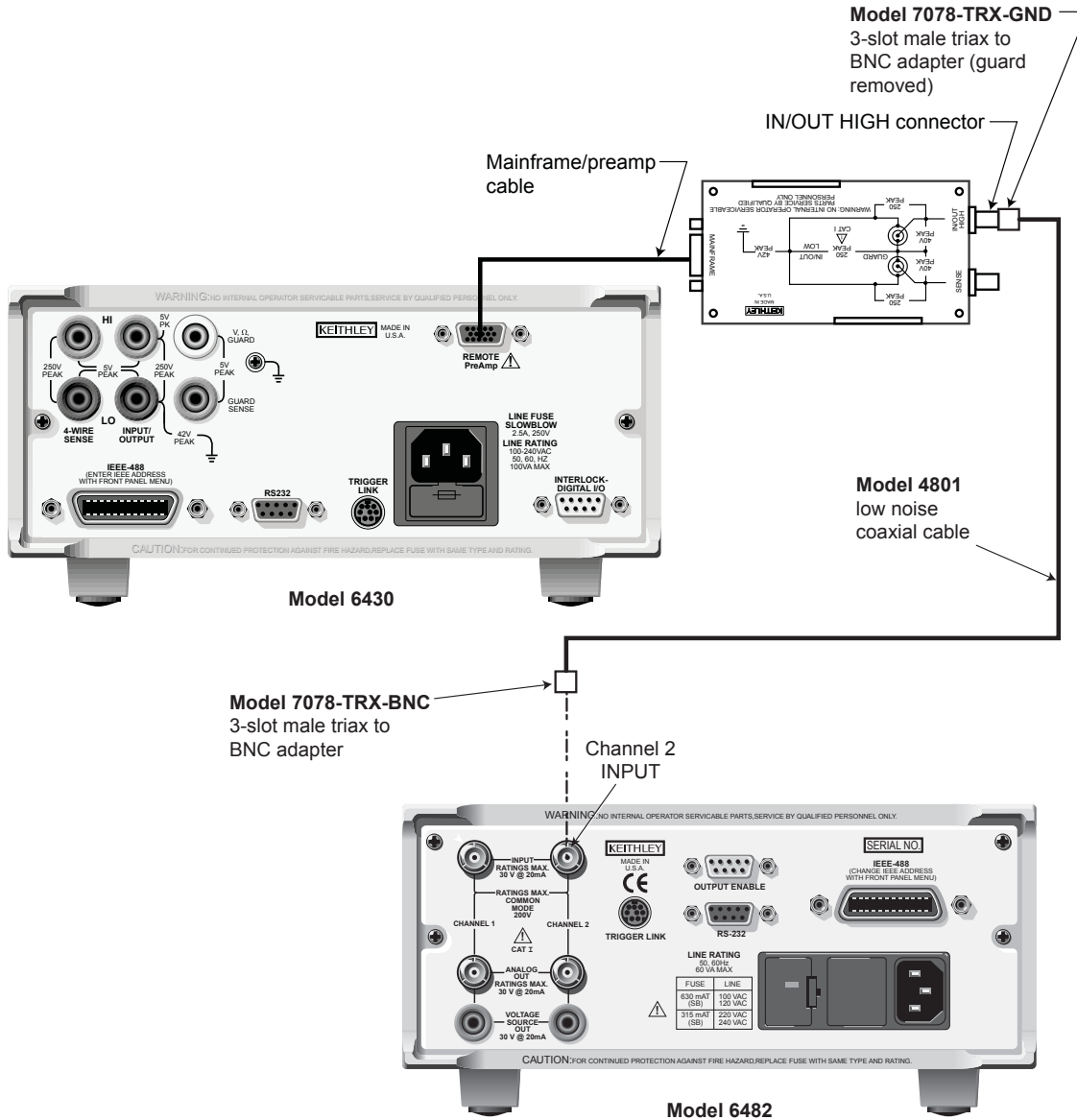
Figure 17-12
Channel 2 current calibration connections (20 µA to 20 mA ranges)
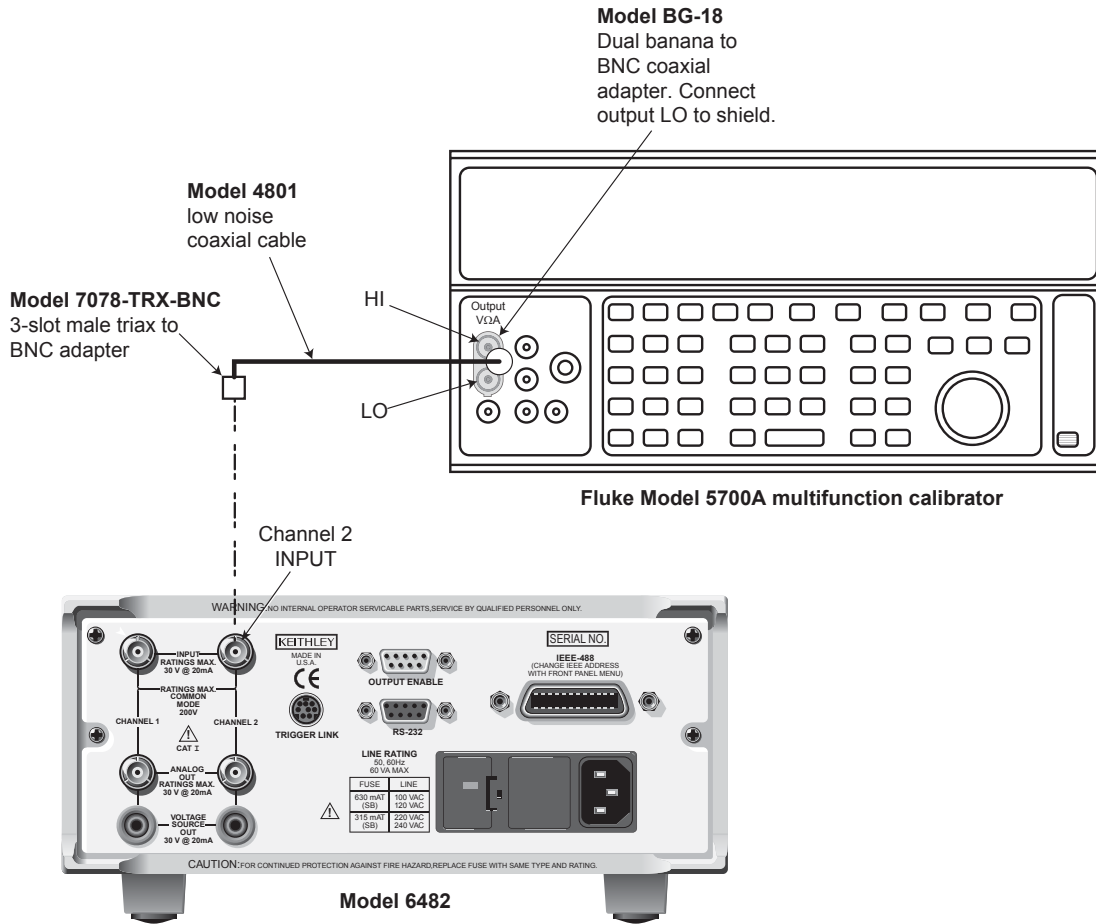
Step 7. Enter calibration adjustment dates and save calibration

NOTE   *For temporary calibration without saving new calibration constants, proceed to Step 8. Lock out calibration.*

1. From the CALIBRATION menu, select **SAVE**, and then press the **ENTER** key. The instrument will prompt you for the calibration date:

   ```
   CAL DATE: 06/01/2012
   Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
   ```

2. Using the EDIT keys, change the displayed date to today's date and press the **ENTER** key. Press the **ENTER** key again to confirm the date.

3. The instrument will then prompt for the calibration due date:

   ```
   NEXT CAL: 06/01/2012
   Use ◀ , ▶ , ▲ , ▼ , ENTER or EXIT.
   ```

4. Set the calibration due date and press the **ENTER** key. Press the **ENTER** key again to confirm the date.

5. When the calibration dates are entered, calibration is complete and the following message will be displayed:

   ```
   CALIBRATION COMPLETE
   Press ENTER to save; EXIT to abort
   ```

6. Press the **ENTER** key to save the calibration data or press the **EXIT** key to abort without saving calibration data.

Step 8. Lock out calibration

From the CAL EXECUTION menu, select **LOCK** and press the **ENTER** key to lock out calibration. Press the **EXIT** key to return to normal display.

Remote calibration

Use the following procedure to perform remote calibration by sending SCPI commands over the IEEE-488 bus or RS-232 port. The remote commands and appropriate parameters are separately summarized for each step.

Remote calibration command summary

Table 17-10 summarizes only those remote calibration commands used in this section.

NOTE   *For a detailed description of all calibration commands and queries, refer to Appendix B.*

Table 17-10
Remote calibration command summary

Command	Description
`:CALibration`	Calibration subsystem.
`:PROTected`	Calibration commands protected by password.
`:CODE '<password>'`	Unlock calibration. Default password: KI006482.
`:SENSe[1] <NRf>`	Calibrate active range of channel 1 current measurement.
`:SENSe2 <NRf>`	Calibrate active range of channel 2 current measurement.
`:SOURce[1] <NRf>`	Calibrate active channel 1 voltage bias range.
`:OFFSet`	Calibrate channel 1 input voltage offset.
`:SOURce2 <NRf>`	Calibrate active channel 2 voltage bias range.
`:OFFSet`	Calibrate channel 2 input voltage offset.
`:DATE <yyyy>,<mm>,<dd>`	Program calibration year, month, day.
`:NDUE <yyyy>,<mm>,<dd>`	Program calibration due year, month, day.
`:SAVE`	Save calibration data in EEPROM.
`:LOCK`	Lock out calibration.

Remote calibration procedure

Step 1. Prepare the Model 6482 for calibration adjustment

1. With the power off, connect the Model 6482 to the controller IEEE-488 interface or RS-232 port using a shielded interface cable.
2. Turn on the Model 6482 and the test equipment, and allow them to warm up for at least one hour before performing calibration.
3. If you are using the IEEE-488 interface, make sure the primary address of the Model 6482 is the same as the address specified in the program you will be using to send commands. Use the **MENU** key and the COMMUNICATION menu to access the IEEE-488 address.
4. Send the following command to unlock calibration:

```
:CAL:PROT:CODE   'KI006482'
```

5. Table 17-11 summarizes the various calibration steps and associated commands, which are covered in more detail throughout the procedure.

Table 17-11
Remote calibration step summary

Calibrated function	Calibration command*	Test connections
Input offset (channel 1)	`:CAL:PROT:SOUR1:OFFS`	Triaxial shield cap on channel 1 INPUT connector.

Voltage bias (channel 1)	`:OUTP1 ON`
	`:SOUR1:VOLT:RANG <Range>`
	`:SOUR1:VOLT <+FS>`
	`:CAL:PROT:SOUR1 <DMM_Reading>`
	`:SOUR1:VOLT 0`
	`:CAL:PROT:SOUR1 <DMM_Reading>`
	`:SOUR1:VOLT <-FS>`
	`:CAL:PROT:SOUR1 <DMM_Reading>`
	`:OUTP1 OFF`
	`:OUTP2 ON`
	`:SOUR2:VOLT:RANG <Range>`
	`:SOUR2:VOLT <+FS>`
		DMM to channel 1 OUTPUT/INPUT LO (Figure 17-7).
		
		DMM to channel 1 OUTPUT/INPUT LO (Figure 17-7).
		
		DMM to channel 1 OUTPUT/INPUT LO (Figure 17-7).
		
		DMM to channel 2 OUTPUT/INPUT LO (Figure 17-8).

Input offset (channel 2)	`:CAL:PROT:SOUR2:OFFS`	Triaxial shield cap on channel 2 INPUT connector.

Voltage bias (channel 2)	`:CAL:PROT:SOUR2 <DMM_Reading>`
	`:SOUR2:VOLT 0`
	`:CAL:PROT:SOUR2 <DMM_Reading>`
	`:SOUR2:VOLT <-FS>`
	`:CAL:PROT:SOUR2 <DMM_Reading>`
	`:OUTP2 OFF`
		DMM to channel 2 OUTPUT/INPUT LO (Figure 17-8).
		
		DMM to channel 2 OUTPUT/INPUT LO (Figure 17-8).

Current (channel 1, 2 nA to 2 µA ranges)	`:SENS1:CURR:RANG <Range>`
	`:CAL:PROT:SENS1 <+FS_current>`
	`:CAL:PROT:SENS1 <Zero_current>`
	`:CAL:PROT:SENS1 <-FS_current>`
		Source to channel 1 INPUT (Figure 17-9).
		Source to channel 1 INPUT (Figure 17-9).
		Source to channel 1 INPUT (Figure 17-9).

Current (channel 1, 20 µA to 20 mA ranges)	`:SENS1:CURR:RANG <Range>`
	`:CAL:PROT:SENS1 <+FS_current>`
	`:CAL:PROT:SENS1 <Zero_current>`
	`:CAL:PROT:SENS1 <-FS_current>`
		Source to channel 1 INPUT (Figure 17-10).
		Source to channel 1 INPUT (Figure 17-10).
		Source to channel 1 INPUT (Figure 17-10).

Current (channel 2, 2 nA to 2 µA ranges)	`:SENS2:CURR:RANG <Range>`
	`:CAL:PROT:SENS2 <+FS_current>`
	`:CAL:PROT:SENS2 <Zero_current>`
	`:CAL:PROT:SENS2 <-FS_current>`
		Source to channel 2 INPUT (Figure 17-11).
		Source to channel 2 INPUT (Figure 17-11).
		Source to channel 2 INPUT (Figure 17-11).

Current (channel 2, 20 µA to 20 mA ranges)	`:SENS2:CURR:RANG <Range>`
	`:CAL:PROT:SENS2 <+FS_current>`
	`:CAL:PROT:SENS2 <Zero_current>`
	`:CAL:PROT:SENS2 <-FS_current>`
		Source to channel 2 INPUT (Figure 17-12).
		Source to channel 2 INPUT (Figure 17-12).
		Source to channel 2 INPUT (Figure 17-12).

*Procedure repeated separately for each range. Voltage <Range> = 10 or 30. Current <Range> = 2e-9 to 20e-3 in decade steps.

Step 2. Channel 1 input offset voltage calibration adjustment

1. Install a triaxial shielding cap on the channel 1 INPUT connector.
2. Send this command to calibrate channel 1 input offset voltage:

    `:CAL:PROT:SOUR1:OFFS`

Step 3. Channel 1 voltage source calibration adjustment

Follow these steps to calibrate the channel 1 voltage bias source. Table 17-12 summarizes channel 1 calibration voltages and commands.

1. Connect the DMM to the Model 6482 channel 1 OUTPUT and INPUT LO terminals, as shown in Figure 17-7.
2. Select the DMM DC voltage function and then enable autorange.
3. Send the following command to turn on the channel 1 output:

    `:OUTP1 ON`

4. Send the following command to select the channel 1, 10 V voltage bias range:

    `:SOUR1:VOLT:RANG 10`

5. Send the following command to output +10 V:

    `:SOUR1:VOLT 10`

6. Note and record the DMM reading, and send that value as the parameter for the following command:

    `:CAL:PROT:SOUR1 <DMM_Reading>`

    For example, if the DMM reading is 9.95 V, the correct command is:

    `:CAL:PROT:SOUR1 9.95`

7. Send the following command to output 0 V:

    `:SOUR1:VOLT 0`

8. Note and record the DMM reading, and send that value as the parameter for the following command:

    `:CAL:PROT:SOUR1 <DMM_Reading>`

9. Send the following command to output -10 V:

    `:SOUR1:VOLT -10`

10. Note and record the DMM reading, and send that value as the parameter for the following command:

    `:CAL:PROT:SOUR1 <DMM_Reading>`

11. Repeat steps 4 through 10 for the 30 V range using Table 17-12 as a guide. Be sure to:
    •   Select the 30 V range using the :SOUR1:VOLT:RANG 30 command.
    •   Send the ±30 V source values, where appropriate.
12. Send the following command to turn off the channel 1 output:

    `:OUTP1 OFF`

Table 17-12
Channel 1 remote voltage bias calibration summary

Voltage bias range	Calibration voltages	Calibration commands
10 V		`:OUTP1 ON` `:SOUR1:VOLT:RANG 10`
	+10 V	`:SOUR1:VOLT 10` `:CAL:PROT:SOUR1 <DMM_Reading>`
	0 V	`:SOUR1:VOLT 0` `:CAL:PROT:SOUR1 <DMM_Reading>`
	-10 V	`:SOUR1:VOLT -10` `:CAL:PROT:SOUR1 <DMM_Reading>`
30 V		`:SOUR1:VOLT:RANG 30`
	+30 V	`:SOUR1:VOLT 30` `:CAL:PROT:SOUR1 <DMM_Reading>`
	0 V	`:SOUR1:VOLT 0` `:CAL:PROT:SOUR1 <DMM_Reading>`

	-30 V	`:SOUR1:VOLT -30`
		`:CAL:PROT:SOUR1 <DMM_Reading>`
		`:OUTP1 OFF`

Step 4. Channel 2 input offset voltage calibration adjustment

1. Install a triaxial shielding cap on the channel 2 INPUT connector.
2. Send this command to calibrate channel 2 input offset voltage:

   `:CAL:PROT:SOUR2:OFFS`

Step 5. Channel 2 voltage source calibration adjustment

Follow these steps to calibrate the channel 2 voltage bias source. Table 17-13 summarizes channel 1 calibration voltages and commands.

1. Connect the DMM to the Model 6482 channel 2 OUTPUT and INPUT LO terminals, as shown in Figure 17-8.
2. Select the DMM DC voltage function, and enable autorange.
3. Send this command to turn on the channel 2 output:

   `:OUTP2 ON`

4. Send the following command to select the channel 2, 10 V voltage bias range:

   `:SOUR2:VOLT:RANG 10`

5. Send the following command to output +10 V:

   `:SOUR2:VOLT 10`

6. Note and record the DMM reading, and send that value as the parameter for the following command:

   `:CAL:PROT:SOUR2 <DMM_Reading>`

7. Send the following command to output 0 V:

   ```
   :SOUR2:VOLT 0
   ```

8. Note and record the DMM reading, and then send that value as the parameter for the following command:

   ```
   :CAL:PROT:SOUR2 <DMM_Reading>
   ```

9. Send the following command to output -10 V:

   ```
   :SOUR2:VOLT -10
   ```

10. Note and record the DMM reading, and send that value as the parameter for the following command:

    ```
    :CAL:PROT:SOUR2 <DMM_Reading>
    ```

11. Repeat steps 4 through 10 for the 30 V range using Table 17-13 as a guide. Be sure to:
    • Select the 30 V range using the :SOUR2:VOLT:RANG 30 command.
    • Send the ±30 V source values where appropriate.

12. Send this command to turn off the channel 1 output:

    ```
    :OUTP2 OFF
    ```

Table 17-13
Channel 2 remote voltage bias calibration summary

Voltage bias range	Calibration voltages	Calibration commands

10 V	+10 V
	
	0 V
	
	-10 V
		`:OUTP2 ON`
		`:SOUR2:VOLT:RANG 10`
		`:SOUR2:VOLT 10`
		`:CAL:PROT:SOUR2 <DMM_Reading>`
		`:SOUR2:VOLT 0`
		`:CAL:PROT:SOUR2 <DMM_Reading>`
		`:SOUR2:VOLT -10`
		`:CAL:PROT:SOUR2 <DMM_Reading>`

30 V	+30 V
	
	0 V
	
	-30 V
		`:SOUR2:VOLT:RANG 30`
		`:SOUR2:VOLT 30`
		`:CAL:PROT:SOUR2 <DMM_Reading>`
		`:SOUR2:VOLT 0`
		`:CAL:PROT:SOUR2 <DMM_Reading>`
		`:SOUR2:VOLT -30`
		`:CAL:PROT:SOUR2 <DMM_Reading>`
		`:OUTP2 OFF`

Step 6. Channel 1 current calibration adjustment

Follow these steps to calibrate channel 1 current measurements. Table 17-14 summarizes channel calibration currents and commands.

Table 17-14
Calibration currents and commands

Connections	Current range	Model 6430 calibration currents	Model 5700A calibration currents	Calibration commands

Channel 1:
Figure 17-9
Channel 2:
Figure 17-11
	2 nA	+1.90000 nA
		0.00000 nA
		-1.90000 nA
			-
			-
			-
				`:SENS1:CURR:RANG 2e-9`
				`:CAL:PROT:SENS1 1.9e-9`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-9`

	20 nA	+19.0000 nA
		0.0000 nA
		-19.0000 nA
			-
			-
			-
				`:SENS1:CURR:RANG 2e-8`
				`:CAL:PROT:SENS1 1.9e-8`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-8`

	200 nA	+190.000 nA
		0.000 nA
		-190.000 nA
			-
			-
			-
				`:SENS1:CURR:RANG 2e-7`
				`:CAL:PROT:SENS1 1,9e-7`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-7`

	2 µA	+1.90000µA
		0.00000µA
		-1.90000µA
			-
			-
			-
				`:SENS1:CURR:RANG 2e-6`
				`:CAL:PROT:SENS1 1.9e-6`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-6`

Channel 1:
Figure 17-10
Channel 2:
Figure 17-12
	20 µA	-
		-
		-
			+19.0000µA
			0.0000µA
			-19.0000µA
				`:SENS1:CURR:RANG 2e-5`
				`:CAL:PROT:SENS1 1.9e-5`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-5`

	200 µA	-
		-
		-
			+190.000 µA
			0.000 µA
			-190.000 µA
				`:SENS1:CURR:RANG 2e-4`
				`:CAL:PROT:SENS1 1.9e-4`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-4`

	2 mA	-
		-
		-
			+1.90000 mA
			0.0000 mA
			-1.90000 mA
				`:SENS1:CURR:RANG 2e-3`
				`:CAL:PROT:SENS1 1.9e-3`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-3`

	20 mA	-
		-
		-
			+19.0000 mA
			0.0000 mA
			-19.0000 mA
				`:SENS1:CURR:RANG 2e-2`
				`:CAL:PROT:SENS1 1.9e-2`
				`:CAL:PROT:SENS1 0`
				`:CAL:PROT:SENS1 -1.9e-2`

1. Connect the Model 6430 to the Model 6482 channel 1 INPUT connector, as shown in Figure 17-9.
2. Select the Model 6430 V/R amperes function and set the output current to 1.90000 nA, and then turn on the Model 6430 output.
3. Send the following command to select the channel 1, 2 nA range:
    `:SENS1:CURR:RANG 2e-9`
4. Send the following command to calibrate the positive full-range current point:
    `:CAL:PROT:SENS1 1.9e-9`
5. Set the Model 6430 current output to 0 nA.
6. Send the following command to calibrate the zero current point:
    `:CAL:PROT:SENS1 0`
7. Set the Model 6430 current to -1.90000 nA.

8. Send the following command to calibrate the negative full-range current point:

   `:CAL:PROT:SENS1 -1.9e-9`

9. Repeat steps 3 through 8 for the 20 nA to 2 μA ranges using Table 17-14 as a guide. Be sure to:
   - Select the appropriate range using the :SENS1:CURR:RANG <Range> command.
   - Send the appropriate calibration current values with the :CAL:PROT:SENS1 <Current> command.

10. Connect the Model 5700A to the Model 6482 channel 1 INPUT connector, as shown in Figure 17-10.

11. Select the Model 5700A V/R amperes function and set the output current to 1.90000 μA, and then turn on the Model 5700A output.

12. Send the following command to select the channel 1, 20 μA range:

    `:SENS1:CURR:RANG 2e-5`

13. Send the following command to calibrate the positive full-range current point:

    `:CAL:PROT:SENS1 1.9e-5`

14. Set the Model 5700A current output to 0 μA.

15. Send the following command to calibrate the zero current point:

    `:CAL:PROT:SENS1 0`

16. Set the Model 5700A current to -19.0000 μA.

17. Send the following command to calibrate the negative full-range current point:

    `:CAL:PROT:SENS1 -1.9e-5`

18. Repeat steps 11 through 17 for the 200 μA to 20 mA ranges using Table 17-14 as a guide. Be sure to:
    - Select the appropriate range using the :SENS1:CURR:RANG <Range> command.
    - Send the appropriate calibration current values with the :CAL:PROT:SENS1 <Current> command.

Step 7. Channel 2 current calibration adjustment

Follow these steps to calibrate channel 2 current measurements. Table 17-14 summarizes channel calibration currents and commands.

1. Connect the Model 6430 to the Model 6482 INPUT channel 2 connector, as shown in Figure 17-11.

2. Select the Model 6430 V/R amperes function and set the output current to 1.90000 nA, and then turn on the Model 6430 output.

3. Send the following command to select the channel 2, 2 nA range:

   `:SENS2:CURR:RANG 2e-9`

4. Send the following command to calibrate the positive full-range current point:

   `:CAL:PROT:SENS2 1.9e-9`

5. Set the Model 6430 current output to 0 nA.

6. Send the following command to calibrate the zero current point:

   `:CAL:PROT:SENS2 0`

7. Set the Model 6430 current to -1.90000 nA.

8. Send the following command to calibrate the negative full-range current point:

   `:CAL:PROT:SENS2 -1.9e-9`

9. Repeat steps 3 through 8 for the 20 nA to 2 µA ranges using Table 17-14 as a guide. Be sure to:
    - Select the appropriate range using the :SENS2:CURR:RANG <Range> command.
    - Send the appropriate calibration current values with the :CAL:PROT:SENS1 <Current> command.

10. Connect the Model 5700A to the Model 6482 channel 2 INPUT connector, as shown in Figure 17-10.

11. Select the Model 5700A V/R amperes function and set the output current to 1.90000 µA, and then turn on the Model 5700A output.

12. Send the following command to select the channel 2, 20 µA range:

    `:SENS2:CURR:RANG 2e-5`

13. 13. Send the following command to calibrate the positive full-range current point:

    `:CAL:PROT:SENS2 1.9e-5`

14. Set the Model 5700A current output to 0 µA.

15. Send the following command to calibrate the zero current point:

    `:CAL:PROT:SENS2 0`

16. Set the Model 5700A current to -19.0000 µA.

17. Send the following command to calibrate the negative full-range current point:

    `:CAL:PROT:SENS2 -1.9e-5`

18. Repeat steps 11 through 17 for the 200 µA to 20 mA ranges using Table 17-14 as a guide. Be sure to:
    - Select the appropriate range using the :SENS2:CURR:RANG <Range> command.
    - Send the appropriate calibration current values with the :CAL:PROT:SENS2 <Current> command.

Step 8. Program calibration dates

Use following commands to set the calibration date and calibration due date:

`:CAL:PROT:DATE <yyyy>, <mm>, <dd>`   **Calibration date**

`:CAL:PROT:NDUE <yyyy>, <mm>, <dd>`   **Next calibration due date**

Note that the year, month, and date must be separated by commas.

Step 9. Save calibration constants

Calibration is now complete, so you can store the calibration constants in EEROM by sending the following command:

`:CAL:PROT:SAVE`

NOTE   *Calibration will be temporary unless you send the SAVE command.*

Step 10. Lock out calibration

To lock out further calibration, send the following command after completing the calibration procedure:

`:CAL:PROT:LOCK`

Routine Maintenance

In this section:

Introduction

The information in this section deals with routine maintenance of the Keithley Instruments Model 6482 that can be performed by the operator.

Setting the line voltage and replacing the line fuse

WARNING   ***Before changing the line voltage or replacing the line fuse, disconnect the line cord at the rear panel and remove all test leads connected to the instrument. Failure to do so could expose the operator to hazardous voltages that could result in personal injury or death.***

The Model 6482 operates from a line voltage in the range of 100 V, 120 V, 220 V, and 240 V (±10 percent) at a frequency of 50 Hz or 60 Hz. Before plugging in the instrument, make sure the line voltage setting that is visible through the small window in the power module (see item 4 in Figure 18-1) is correct for the line voltage in your area.

WARNING   ***To prevent injury, death, or instrument damage, use only the correct fuse type (see Table 18-1).***

The power line fuse is accessible from the rear panel and is integral with the AC power module (Figure 18-1).

To change the line voltage setting or replace the line fuse:

1. Place the tip of a flat-blade screwdriver into the power module by the fuse holder assembly (see item 1 in Figure 18-1). Gently push in and to the left. Release pressure on the assembly (2) and its internal spring will push it out of the power module.
2. Remove the fuse (3) and replace it with the type listed in Table 18-1.

CAUTION   **For continued protection against fire or instrument damage, replace the fuse only with the type and rating listed. If the power line fuse repeatedly opens (the fuse blows), the instrument will require servicing.**

3. If configuring the instrument for a different line voltage, remove the line voltage selector from the assembly and rotate it to the proper position. When the selector is installed into the fuse holder assembly, the correct line voltage appears inverted in the window (4).
4. Install the fuse holder assembly into the power module by pushing it in until it locks in place.

If the power line fuse continues to open, a circuit malfunction exists and must be corrected. Return the instrument to Keithley Instruments for repair.

Figure 18-1
Model 6482 rear panel line fuse

Model 6482
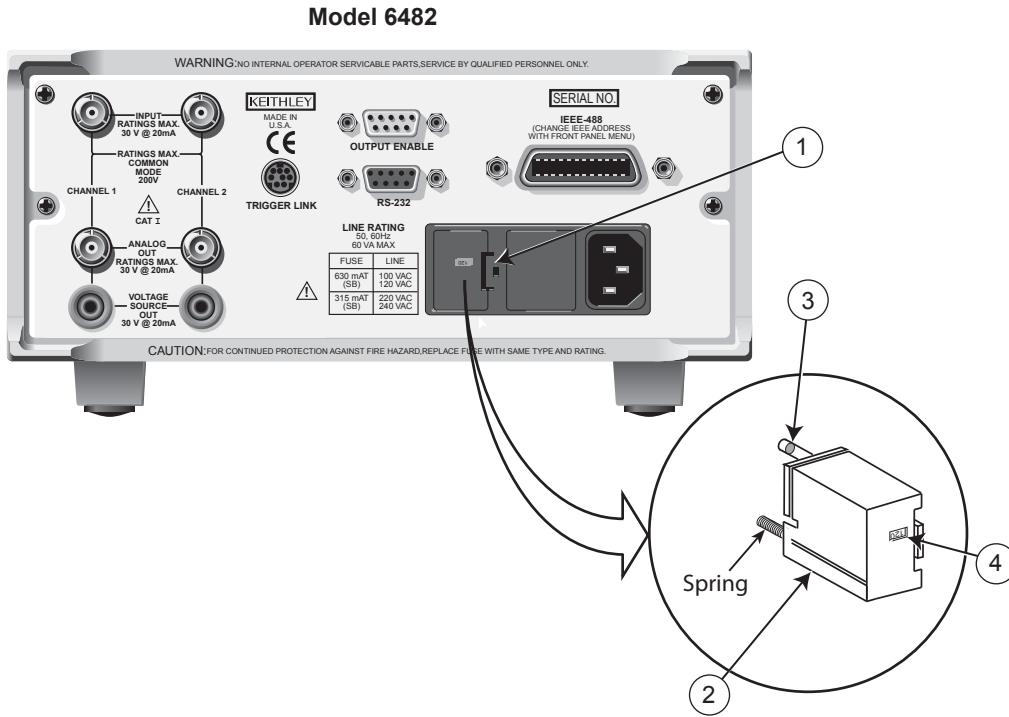
Table 18-1
Power line fuses

Line voltage	Fuse rating	Keithley Instruments part number
100 V, 120 V	0.630 A Slo Blo, 250 V, 5 × 20mm	FU-106-.630
220 V, 240 V	0.315 A Slo Blo, 250 V, 5 × 20mm	FU-106-.315

This page left blank intentionally.

Status and Error Messages

In this appendix:

Introduction

This appendix contains a summary of status and error messages, which status register bits are set when messages occur, and methods to avoid or eliminate most common SCPI errors.

Status and error messages

Table A-1 summarizes status and error messages, which are stored in the Error Queue. Each message is preceded by a code number. Negative (-) numbers are used for SCPI-defined messages, and positive (+) numbers are used for Keithley-defined messages. Note that error and status conditions will also set specific bits in various status registers, as summarized in Table A-1.

Section 13 has detailed information on registers and queues. To summarize, you can use the following queries to obtain error and status information:

- :SYST:ERR? — Reads Error Queue.
- *ESR? — Reads Standard Event Status Register.
- :STAT:OPER? — Reads Operation Event Register.
- :STAT:MEAS? — Reads Measurement Event Register.
- :STAT:QUES? — Reads Questionable Event Register.

NOTE   *SCPI-confirmed messages are described in volume 2: Command Reference of the Standard Commands for Programmable Instruments. Refer to the :SYSTem:ERRor? command.*

Table A-1
Status and error messages

Number	Error message	Event[1]	Status register[2]	Bit
-440	Query UNTERMINATED after indefinite response	EE	Standard Event	2
-430	Query DEADLOCKED	EE	Standard Event	2
-420	Query UNTERMINATED	EE	Standard Event	2
-410	Query INTERRUPTED	EE	Standard Event	2
-363	Input buffer overrun	EE	Standard Event	3
-362	Framing error in program message	EE	Standard Event	3
-361	Parity error in program message	EE	Standard Event	3
-360	Communications error	EE	Standard Event	3
-350	Queue overflow	SYS	Standard Event	3
-330	Self-test failed	EE	Standard Event	3
-314	Save/recall memory lost	EE	Standard Event	3
-315	Configuration memory lost	EE	Standard Event	3
-285	Program syntax error	EE	Standard Event	4
-284	Program currently running	EE	Standard Event	4
-282	Illegal program name	EE	Standard Event	4
-281	Cannot create program	EE	Standard Event	4
-260	Expression error	EE	Standard Event	4
-241	Hardware missing	EE	Standard Event	4
-230	Data corrupt or stale	EE	Standard Event	4
-225	Out of memory	EE	Standard Event	4
-224	Illegal parameter value	EE	Standard Event	4
-223	Too much data	EE	Standard Event	4
-222	Parameter data out of range	EE	Standard Event	4
-221	Settings conflict	EE	Standard Event	4
-220	Parameter error	EE	Standard Event	4
-215	Arm deadlock	EE	Standard Event	4
-214	Trigger deadlock	EE	Standard Event	4
-213	Init ignored	EE	Standard Event	4
-212	Arm ignored	EE	Standard Event	4
-211	Trigger ignored	EE	Standard Event	4
-210	Trigger error	EE	Standard Event	4
-202	Settings lost due to rtl	EE	Standard Event	4
-201	Invalid while in local	EE	Standard Event	4
-200	Execution error	EE	Standard Event	4

[1] EE = Error Event, SE = Status Event, SYS = System Error Event
[2] Use following queries to read status registers:
 Standard Event: *ESR?
 Operation Event: STAT:OPER?
 Measurement Event: STAT:MEAS?
 Questionable Event: STAT:QUES?

Table A-1 (cont.)
Status and error messages

Number	Error message	Event[1]	Status register[2]	Bit
-178	Expression data not allowed	EE	Standard Event	5
-171	Invalid expression	EE	Standard Event	5
-170	Expression error	EE	Standard Event	5
-168	Block data not allowed	EE	Standard Event	5
-161	Invalid block data	EE	Standard Event	5
-160	Block data error	EE	Standard Event	5
-158	String data not allowed	EE	Standard Event	5
-154	String too long	EE	Standard Event	5
-151	Invalid string data	EE	Standard Event	5
-150	String data error	EE	Standard Event	5
-148	Character data not allowed	EE	Standard Event	5
-144	Character data too long	EE	Standard Event	5
-141	Invalid character data	EE	Standard Event	5
-140	Character data error	EE	Standard Event	5
-128	Numeric data not allowed	EE	Standard Event	5
-124	Too many digits	EE	Standard Event	5
-123	Exponent too large	EE	Standard Event	5
-121	Invalid character in number	EE	Standard Event	5
-120	Numeric data error	EE	Standard Event	5
-114	Header suffix out of range	EE	Standard Event	5
-113	Undefined header	EE	Standard Event	5
-112	Program mnemonic too long	EE	Standard Event	5
-111	Header separator error	EE	Standard Event	5
-110	Command header error	EE	Standard Event	5
-109	Missing parameter	EE	Standard Event	5
-108	Parameter not allowed	EE	Standard Event	5
-105	GET not allowed	EE	Standard Event	5
-104	Data type error	EE	Standard Event	5
-103	Invalid separator	EE	Standard Event	5
-102	Syntax error	EE	Standard Event	5
-101	Invalid character	EE	Standard Event	5
-100	Command error	EE	Standard Event	5

[1] EE = Error Event, SE = Status Event, SYS = System Error Event
[2] Use following queries to read status registers:
  Standard Event: *ESR?
  Operation Event: STAT:OPER?
  Measurement Event: STAT:MEAS?
  Questionable Event: STAT:QUES?

Table A-1 (cont.)
Status and error messages

Number	Error message	Event[1]	Status register[2]	Bit
+000	No error	SE		
	Measurement events:			
+100	Limit 1 failed	SE	Measurement Event	0
+101	Low limit 2 failed	SE	Measurement Event	1
+102	High limit 2 failed	SE	Measurement Event	2
+103	Low limit 3 failed	SE	Measurement Event	3
+104	High limit 3 failed	SE	Measurement Event	4
+105	Active limit tests passed	SE	Measurement Event	5
+106	Reading available	SE	Measurement Event	6
+107	Reading overflow	SE	Measurement Event	7
+108	Buffer available	SE	Measurement Event	8
+109	Buffer full	SE	Measurement Event	9
+111	OUTPUT enable asserted	SE	Measurement Event	11
+114	Source in compliance	SE	Measurement Event	14
	Standard events:			
+200	Operation complete	SE	Standard Event	0
	Operation events:			
+300	Device calibrating	SE	Operation Event	0
+303	Device sweeping	SE	Operation Event	3
+305	Waiting in trigger layer	SE	Operation Event	5
+306	Waiting in arm layer	SE	Operation Event	6
+310	Entering idle layer	SE	Operation Event	10
	Questionable events:			
+408	Questionable Calibration	SE	Questionable Event	8
+414	Command Warning	SE	Questionable Event	14

[1] EE = Error Event, SE = Status Event, SYS = System Error Event
[2] Use following queries to read status registers:
  Standard Event: *ESR?
  Operation Event: STAT:OPER?
  Measurement Event: STAT:MEAS?
  Questionable Event: STAT:QUES?

Table A-1 (cont.)
Status and error messages

Number	Error message	Event[1]	Status register[2]	Bit
	Calibration errors:			
+500	Date of calibration not set	EE	Standard Event	3
+501	Next date of calibration not set	EE	Standard Event	3
+502	Calibration data invalid	EE	Standard Event	3
+503	DAC calibration overflow	EE	Standard Event	3
+504	DAC calibration underflow	EE	Standard Event	3
+505	Source offset data invalid	EE	Standard Event	3
+506	Source gain data invalid	EE	Standard Event	3
+507	Measurement offset data invalid	EE	Standard Event	3
+508	Measurement gain data invalid	EE	Standard Event	3
+509	Not permitted with cal locked	EE	Standard Event	3
+510	Not permitted with cal unlocked	EE	Standard Event	3
	Lost data errors:			
+601	Reading buffer data lost	EE	Standard Event	3
+602	GPIB address lost	EE	Standard Event	3
+603	Power-on state lost	EE	Standard Event	3
+604	DC calibration data lost	EE	Standard Event	3
+605	Calibration dates lost	EE	Standard Event	3
+606	GPIB communication language lost	EE	Standard Event	3
	Communication errors:			
+700	Invalid system communication	EE	Standard Event	3
+701	ASCII only with RS-232	EE	Standard Event	3
	Additional command execution errors:			
	Illegal with storage active			
+800	Insufficient vector data	EE	Standard Event	4
+801	OUTPUT blocked by output enable	EE	Standard Event	4
+802	Not permitted with OUTPUT off	EE	Standard Event	4
+803		EE	Standard Event	4

[1] EE = Error Event, SE = Status Event, SYS = System Error Event
[2] Use following queries to read status registers:
  Standard Event: *ESR?
  Operation Event: STAT:OPER?
  Measurement Event: STAT:MEAS?
  Questionable Event: STAT:QUES?

Table A-1 (cont.)
Status and error messages

Number	Error message	Event[1]	Status register[2]	Bit
+822	Too small for sense range	EE	Standard Event	4
+824	Cannot exceed compliance range	EE	Standard Event	4
+830	Invalid with INF ARM:COUNT	EE	Standard Event	4
+900	Internal System Error	EE	Standard Event	3

[1] EE = Error Event, SE = Status Event, SYS = System Error Event
[2] Use following queries to read status registers:
  Standard Event: *ESR?
  Operation Event: STAT:OPER?
  Measurement Event: STAT:MEAS?
  Questionable Event: STAT:QUES?

Eliminating common SCPI errors

There are three SCPI errors that occur more often than any others:

- -113, Undefined header
- -410, Query INTERRUPTED
- -420, Query UNTERMINATED

The following paragraphs discuss the causes for these errors and methods for avoiding them.

-113, Undefined header

This error indicates that the command you sent to the instrument did not contain a recognizable command name. The most likely causes for this error are:

- Missing space between the command and its parameter: There must be one or more spaces (blanks) between the command and its parameter. For example:
  :SENS1:CURR:RANG2e3   Incorrect (no space between command and parameter)
  :SENS1:CURR:RANG 2e3   Correct
- Improper short or long form: Check the command list in Section 16 for the correct command name.
- Blanks (spaces) within the command name: For example:
  :SYST :ERR?   Incorrect (space between :SYST and :ERR?)
  :SYST:ERR?   Correct

- 410, Query INTERRUPTED

This error occurs when you have sent a valid query to the instrument, and then send it to another command or query, or a Group Execute Trigger (GET) before it has had a chance to send the entire response message (including the line-feed/EOI terminator). The most likely causes are:

- Sending a query to the instrument and then sending another command or query before reading the response to the first query: For example, the following sequence of commands will cause an error -410:
  :SYST:ERR?
  *OPC?
  This sequence generates an error because you must read the response to :SYST:ERR? before sending the *OPC? query.
- Incorrectly configured IEEE-488 driver*:* The driver must be configured so that when talking on the bus it sends line-feed with EOI as the terminator, and when listening on the bus it

expects line-feed with EOI as the terminator. See the reference manual for your particular IEEE-488 interface.

-420, Query UNTERMINATED

This error occurs when you address the instrument to talk, and there is no response message to send. The most likely causes are:

- Not sending a query: You must send a valid query to the instrument before addressing it to talk.
- Sending an invalid query: If you have sent a query and still get this error, make sure that the instrument is processing the query without error. For example, sending an ill-formed query that generates an error -113, Undefined header and then addressing the instrument to talk will generate an error -420, Query UNTERMINATED as well.
- Valid query following an invalid command: This situation can occur when you send multiple commands or queries (program message units) within one command string (program message). When the Model 6482 detects an error in a program message unit, it discards all further program message units until the end of the string; for example:

  :SENS1:DATE?; :SOUR1:VOLT?

  In the above program message, the program message unit :SENS1:DATE? will generate error -113, Undefined header, and the Model 6482 will discard the second program message unit :SOUR1:VOLT? even though it is a valid query.

In this appendix:

Introduction

Data flow for remote operation is summarized by the block diagram shown in Figure B-1. Refer to this block diagram for the following discussion.

Figure B-1
Data flow block diagram
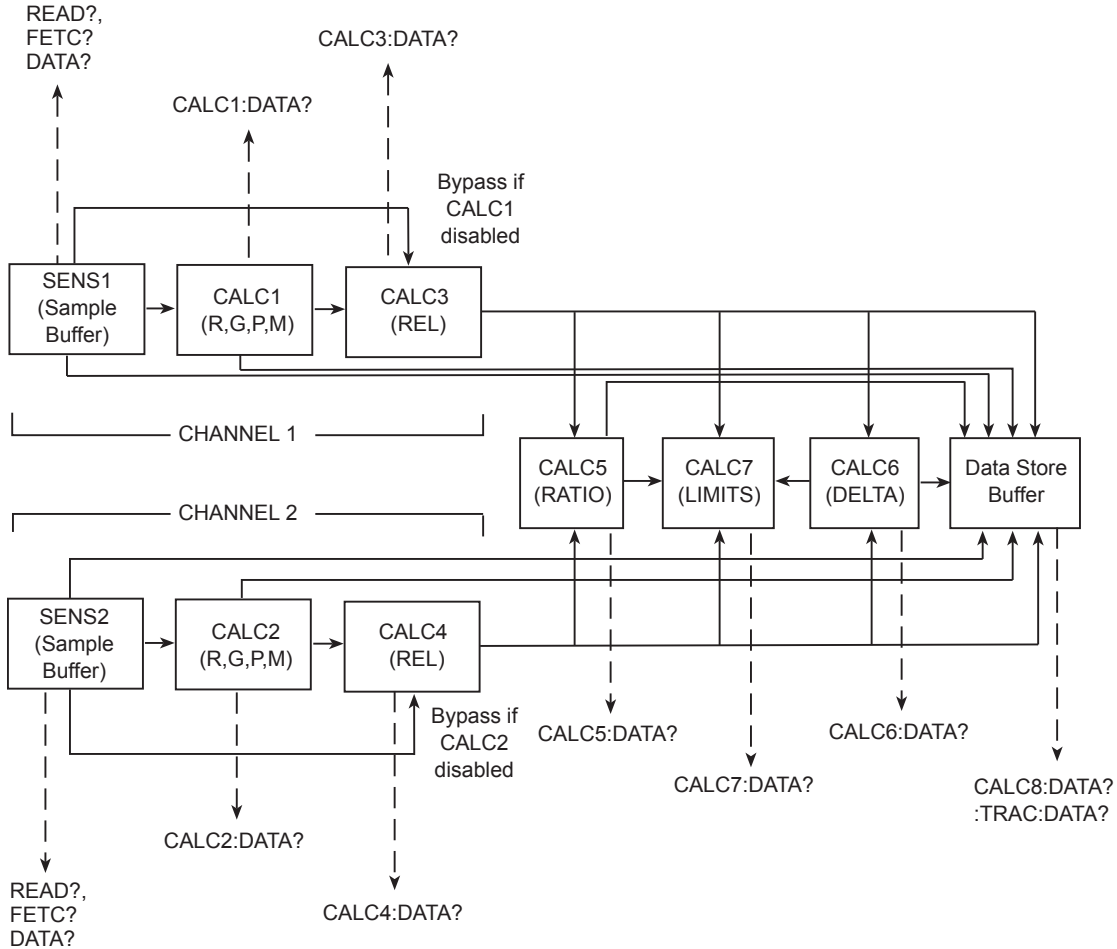
SENS1 and SENS2

The SENS1 and SENS2 blocks represent the basic measured current readings for channel 1 and channel 2 respectively. If the filter is enabled, the readings will be filtered. The SENSE blocks also measure time for the timestamp.

INIT

When the INITiate command is sent, the programmed number of source and measure operations are performed, and the respective data is temporarily stored in a sample buffer. For example, if 20 source and measure operations were performed, then 20 sets of data will be stored in the Buffer. Data from this buffer is then routed to other enabled data flow blocks.

Assuming that all functions are enabled, the data that is output by the read commands (FETCh?, CALCX:DATA?, and TRACe:DATA?) depend on which data elements are selected. With all elements selected, available data will include channel 1 and channel 2 measurements as well as the timestamp and status information. Note that if a channel is not enabled, then either the NAN (not a number) value or the source value will be used instead. See Section 16, "FORMat subsystem," for details.

After all source and measure operations are completed, the Model 6482 returns to the idle state. The data stored in the sample buffer will remain there until data from another source-measure cycle overwrites the buffer. Data in the sample buffer is lost if the Model goes to the local state (REM indicator off).

NOTE   *With no data in the Sample Buffer, the FETCh? and CALCulateX:DATA? commands to read data will display the message Data corrupt or stale.*

FETCh?

This command is used to read data stored in the sample buffer. If, for example, there are 20 data sets stored in the sample buffer, then all 20 data sets will be sent to the computer when FETCh? is executed. Note that FETCh? does not affect data in the sample buffer. Thus, subsequent executions of FETCh? acquire the same data.

READ?

The READ? command performs an INITiate and then a FETCh? The INITiate triggers a new source and measure cycle which puts new data in the sample buffer. FETCh? reads that new data. The MEASure? command places the Model 6482 in a single source and measure mode and then performs a READ?. See Section 15 for more information on READ? and MEASure?.

CALCulate[1]:DATA? and CALCulate2:DATA?

If CALCulate1 or CALCulate2 is enabled, SENS1 or SENS2 data is fed to the CALC1 (channel 1) or CALC2 (channel 2) block where the results for the selected math function are calculated. The CALC1:DATA? or CALC2:DATA? command will read the results of the math function. If CALC1 or CALC2 is enabled, that block is bypassed, and SENS1 or SENS2 data is fed directly to CALC3 or CALC4.

CALCulate3:DATA? and CALCulate4:DATA?

If CALCulate3 or CALCulate4 (channel 1 or channel 2 REL) is enabled, data is fed to the CALC3 (channel 1) or CALC4 (channel 2) block where the programmed or acquired REL value is subtracted from the data. CALC3 data comes from either CALC1 or SENS1, depending on the selected feed. Similarly, CALC4 data comes from CALC2 or SENS2, depending on the selected feed. The CALC3:DATA? or CALC4:DATA? command will read the results of the REL operation.

Note that CALC3 and CALC4 can also be fed to the CALC7 block for limit testing, and to the data store buffer for :TRACe:DATA? CALC8:DATA? buffer statistics.

CALCulate5:DATA? and CALCulate6:DATA?

If CALCulate5 (RATIO) or CALCulate6 (DELTA) is enabled, data is fed to the CALC5 or CALC6 block where the ratio or difference between measurement data for the two channels is calculated. The CALC5:DATA? returns the calculated ratio result, and CALC6:DATA? reads the results of the delta computation. Note that CALC5 and CALC6 can also be fed to the CALC7 block for limit testing, and to the data store buffer for :TRACe:DATA? CALC8:DATA? buffer statistics.

CALCulate7:DATA?

If CALCulate7 is enabled, data becomes available to the CALC7 block for limit testing. Depending on the selected feed, limit testing can be performed on the channel 1 measurement function, channel 2 measurement, RATIO, or DELTA, If null (rel) is enabled, the readings used for limit testing will be the results of the null operation.

The CALCulate7:DATA? command acquires the readings used for limit testing.

TRACe:DATA?

If the data store is enabled, data becomes available to the TRACE block for storage. The selected :FORM:ELEM:TRAC data elements determines which group of readings are stored (CALC3 through CALC6). The TRACe:DATA? command reads the entire contents of the data store.

CALCulate8:DATA?

Statistical information (minimum, maximum, mean, standard deviation, and peak-to-peak) is available for measure readings stored in the buffer. The CALCulate8:DATA? command acquires the results of the selected statistical calculation. The data source for CALC8 is determined by the :DISPlay:MODE command.

IEEE-488 Bus Overview

In this appendix:

Introduction

The IEEE-488 bus is a communication system between two or more electronic devices. A device can be either an instrument or a computer. When a computer is used on the bus, it serves to supervise the communication exchange between all the devices and is known as the controller. Supervision by the controller consists of determining which device will talk and which device will listen. As a talker, a device will output information and as a listener, a device will receive information. To simplify the task of keeping track of the devices, a unique address number is assigned to each one.

On the bus, only one device can talk at a time and is addressed to talk by the controller. The device that is talking is known as the active talker. The devices that need to listen to the talker are addressed to listen by the controller. Each listener is then referred to as an active listener. Devices that do not need to listen are instructed to unlisten. The reason for the unlisten instruction is to optimize the speed of bus information transfer since the task of listening takes up bus time.

Through the use of control lines, a handshake sequence takes place in the transfer process of information from a talker to a listener. This handshake sequence helps ensure the credibility of the information transfer. The basic handshake sequence between an active controller (talker) and a listener is as follows:

1. The listener indicates that it is ready to listen.
2. The talker places the byte of data on the bus and indicates that the data is available to the listener.
3. The listener, aware that the data is available, accepts the data and then indicates that the data has been accepted.
4. The talker, aware that the data has been accepted, stops sending data and indicates that data is not being sent.
5. The listener, aware that there is no data on the bus, indicates that it is ready for the next byte of data.

Bus description

The IEEE-488 bus, which is also frequently referred to as the GPIB (General Purpose Interface Bus), was designed as a parallel transfer medium to optimize data transfer without using an excessive number of bus lines. In keeping with this goal, the bus has only eight data lines that are used for both data and with most commands. Five bus management lines and three handshake lines round out the complement of bus signal lines.

A typical setup for controlled operation is shown in Figure C-1. Generally, a system will contain one controller and a number of other instruments to which the commands are given. Device operation is categorized into three operators: controller, talker, and listener. The controller does what its name implies; it controls the instruments on the bus. The talker sends data while a listener receives data. Depending on the type of instrument, any particular device can be a talker only, a listener only, or both a talker and listener.

There are two categories of controllers: system controller and basic controller. Both are able to control other instruments, but only the system controller has the absolute authority in the system. In a system with more than one controller, only one controller may be active at any given time. Certain protocol is used to pass control from one controller to another.

The IEEE-488 bus is limited to 15 devices, including the controller. Thus, any number of talkers and listeners up to that limit may be present on the bus at one time. Although several devices may be commanded to listen simultaneously, the bus can have only one active talker, or communications would be scrambled.

A device is placed in the talk or listen state by sending an appropriate talk or listen command. These talk and listen commands are derived from an instrument's primary address. The primary address may have any value between 0 and 31, and is generally set by rear panel DIP switches or programmed from the front panel of the instrument. The actual listen address value sent out over the bus is obtained by ORing the primary address with #H20. For example, if the primary address is #H16, the actual listen address is #H36 (#H36 = #H16 + #H20). In a similar manner, the talk address is obtained by ORing the primary address with #H40. With the present example, the talk address derived from a primary address of 16 decimal would be #H56 (#H56 = #H16 + #H40).

The IEEE-488 standards also include another addressing mode called secondary addressing. Secondary addresses lie in the range of #H60-#H7F. Note, however, that many devices, including the Model 6482, do not use secondary addressing.

Once a device is addressed to talk or listen, the appropriate bus transactions take place. For example: if the instrument is addressed to talk, it places its data string on the bus one byte at a time. The controller reads the information and the appropriate software can be used to direct the information to the correct location.

Figure C-1
IEEE-488 bus configuration
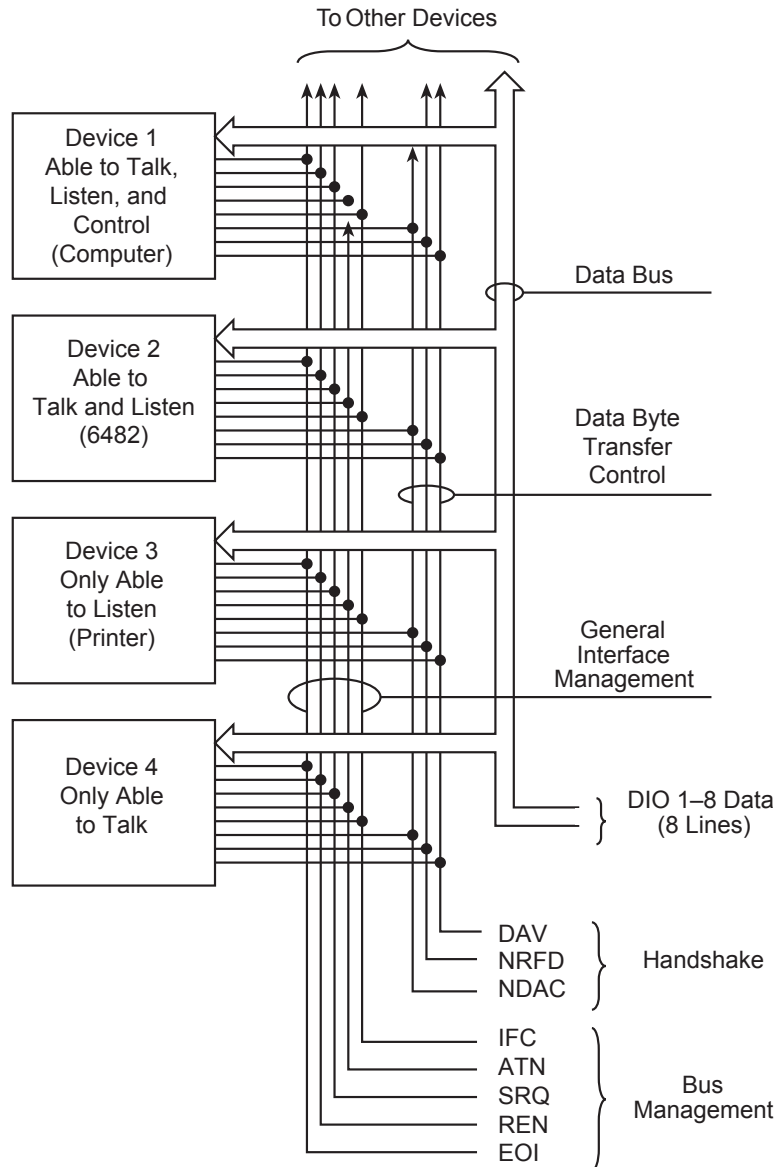
Bus lines

The signal lines on the IEEE-488 bus are grouped into three different categories: data lines, management lines, and handshake lines. The data lines handle bus data and commands, while the management and handshake lines ensure that proper data transfer and operation takes place. Each bus line is active low, with approximately zero volts representing a logic 1 (true). The following paragraphs describe the operation of these lines.

Data lines

The IEEE-488 bus uses eight data lines that transfer data one byte at a time. DIO1 (Data Input/ Output) through DIO8 (Data Input/Output) are the eight data lines used to transmit both data and multiline commands and are bidirectional. The data lines operate with low true logic.

Bus management lines

The five bus management lines help to ensure proper interface control and management. These lines are used to send the uniline commands.

ATN (Attention): The ATN line is one of the more important management lines. The state of this line determines how information on the data bus is to be interpreted.

IFC (Interface Clear): As the name implies, the IFC line controls clearing of instruments from the bus.

REN (Remote Enable): The REN line is used to place the instrument on the bus in the remote mode.

EOI (End or Identify): The EOI is usually used to mark the end of a multi-byte data transfer sequence.

SRQ (Service Request): This line is used by devices when they require service from the controller.

Handshake lines

The bus handshake lines operate in an interlocked sequence. This method ensures reliable data transmission regardless of the transfer rate. Generally, data transfer will occur at a rate determined by the slowest active device on the bus.

One of the three handshake lines is controlled by the source (the talker sending information), while the remaining two lines are controlled by accepting devices (the listener or listeners receiving the information). The three handshake lines are:

DAV (DATA VALID): The source controls the state of the DAV line to indicate to any listening devices whether or not data bus information is valid.

NRFD (Not Ready For Data): The acceptor controls the state of NRFD. It is used to signal to the transmitting device to hold off the byte transfer sequence until the accepting device is ready.

NDAC (Not Data Accepted): NDAC is also controlled by the accepting device. The state of NDAC tells the source whether or not the device has accepted the data byte.

The complete handshake sequence for one data byte is shown in Figure C-2. Once data is placed on the data lines, the source checks to see that NRFD is high, indicating that all active devices are ready. At the same time, NDAC should be low from the previous byte transfer. If these conditions are not met, the source must wait until NDAC and NRFD have the correct status. If the source is a controller, NRFD and NDAC must be stable for at least 100 ns after ATN is set true. Because of the possibility of a bus hang up, many controllers have time-out routines that display messages in case the transfer sequence stops for any reason.

Once all NDAC and NRFD are properly set, the source sets DAV low, indicating to accepting devices that the byte on the data lines is now valid. NRFD will then go low, and NDAC will go high once all devices have accepted the data. Each device will release NDAC at its own rate, but NDAC will not be released to go high until all devices have accepted the data byte.

The sequence just described is used to transfer both data, talk, and listen addresses, as well as multiline commands. The state of the ATN line determines whether the data bus contains data, addresses, or commands as described in "Bus commands," page C-6.

Figure C-2
IEEE-488 handshake sequence
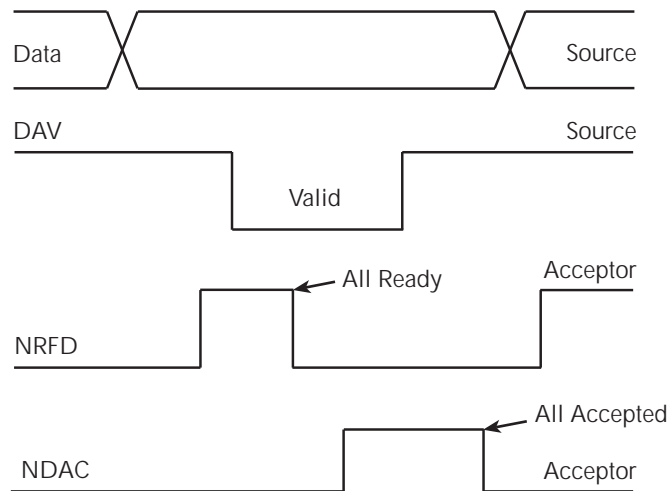
Bus commands

The instrument may be given a number of special bus commands through the IEEE-488 interface. This section briefly describes the purpose of the bus commands which are grouped into the following four categories.

1. **Uniline commands:** Sent by setting the associated bus lines true. For example, to assert REN (Remote Enable), the REN line would be set low (true).
2. **Multiline commands:** General bus commands which are sent over the data lines with the ATN line true (low).
3. **Common commands:** Commands that are common to all devices on the bus; sent with ATN high (false).
4. **SCPI commands**: Commands that are particular to each device on the bus; sent with ATN (false).

These bus commands and their general purpose are summarized in Table C-1.

Table C-1
IEEE-488 bus command summary

Command type	Command	State of ATN line	Comments
Uniline	REN (Remote Enable)	X	Set up devices for remote operation.
	EOI (End Or Identify)	X	Marks end of transmission.
	IFC (Interface Clear)	X	Clears interface.
	ATN (Attention)	Low	Defines data bus contents.
	SRQ (Service Request)	X	Controlled by external device.
Multiline Universal	LLO (Local Lockout)	Low	Locks out local operation.
	DCL (Device Clear)	Low	Returns device to default conditions.
	SPE (Serial Poll Enable)	Low	Enables serial polling.
	SPD (Serial Poll Disable)	Low	Disables serial polling.
Addressed	SDC (Selective Device Clear)	Low	Returns instrument to default conditions.
	GTL (Go To Local)	Low	Returns device to local.
Unaddressed	UNL (Unlisten)	Low	
	UNT (Untalk)	Low	Removes all listeners from the bus. Removes any talkers from the bus.
Common	–	High	
SCPI	–	High	Programs IEEE-488.2 compatible instruments for common operations. Programs SCPI compatible instruments for particular operations.

Uniline commands

ATN, IFC, and REN are asserted only by the controller. SRQ is asserted by an external device. EOI may be asserted either by the controller or other devices depending on the direction of data transfer. The following is a description of each command. Each command is sent by setting the corresponding bus line true.

REN (Remote Enable): REN is sent to set up instruments on the bus for remote operation. When REN is true, devices will be removed from the local mode. Depending on device configuration, all front panel controls except the LOCAL key (if the device is so equipped) may be locked out when REN is true. Generally, REN should be sent before attempting to program instruments over the bus.

EOI (End or Identify): EOI is used to positively identify the last byte in a multi-byte transfer sequence, thus allowing data words of various lengths to be transmitted easily.

IFC (Interface Clear): IFC is used to clear the interface and return all devices to the talker and listener idle states.

ATN (Attention): The controller sends ATN while transmitting addresses or multiline commands.

SRQ (Service Request): SRQ is asserted by a device when it requires service from a controller.

Universal multiline commands

Universal commands are those multiline commands that require no addressing. All devices equipped to implement such commands will do so simultaneously when the commands are transmitted. As with all multiline commands, these commands are transmitted with ATN true.

LLO (Local Lockout): LLO is sent to the instrument to lock out the LOCAL key and thus all the front panel controls.

DCL (Device Clear): DCL is used to return instruments to some default state. Usually, instruments return to the power-up conditions.

SPE (Serial Poll Enable): SPE is the first step in the serial polling sequences, which is used to determine which device has requested service.

SPD (Serial Poll Disable): SPD is used by the controller to remove all devices on the bus from the serial poll mode and is generally the last command in the serial polling sequence.

Addressed multiline commands

Addressed commands are multiline commands that must be preceded by the device listen address before that instrument will respond to the command in question. Note that only the addressed device will respond to these commands. Both the commands and the address preceding it are sent with ATN true.

SDC (Selective Device Clear): The SDC command performs essentially the same function as the DCL command except that only the addressed device responds. Generally, instruments return to their power-up default conditions when responding to the SDC command.

GTL (Go To Local): The GTL command is used to remove instruments from the remote mode. With some instruments, GTL also unlocks front panel controls if they were previously locked out with the LLO command.

GET (Group Execute Trigger): The GET command is used to trigger devices to perform a specific action that depends on device configuration (for example, take a reading). Although GET is an addressed command, many devices respond to GET without addressing.

Address commands

Addressed commands include two primary command groups and a secondary address group. ATN is true when these commands are asserted. The commands include:

LAG (Listen Address Group): These listen commands are derived from an instrument's primary address and are used to address devices to listen. The actual command byte is obtained by ORing the primary address with #H20.

TAG (Talk Address Group): The talk commands are derived from the primary address by ORing the address with #H40. Talk commands are used to address devices to talk.

SCG (Secondary Command Group): Commands in this group provide additional addressing capabilities. Many devices (including the Model 6482) do not use these commands.

Unaddress commands

The two unaddress commands are used by the controller to remove any talkers or listeners from the bus. ATN is true when these commands are asserted.

UNL (Unlisten): Listeners are placed in the listener idle state by the UNL command.

UNT (Untalk): Any previously commanded talkers will be placed in the talker idle state by the UNT command.

Common commands

Common commands are commands that are common to all devices on the bus. These commands are designated and defined by the IEEE-488.2 standard.

Generally, these commands are sent as one or more ASCII characters that tell the device to perform a common operation, such as reset. The IEEE-488 bus treats these commands as data in that ATN is false when the commands are transmitted.

SCPI commands

SCPI commands are commands that are particular to each device on the bus. These commands are designated by the instrument manufacturer, and are based on the instrument model defined by the Standard Commands for Programmable Instruments (SCPI) Consortium's SCPI standard.

Generally, these commands are sent as one or more ASCII characters that tell the device to perform a particular operation, such as setting a range or closing a relay. The IEEE-488 bus treats these commands as data in that ATN is false when the commands are transmitted.

Command codes

Command codes for the various commands that use the data lines are summarized in Figure C-3. Hexadecimal and the decimal values for the various commands are listed in Table C-2.

Table C-2
Hexadecimal and decimal command codes

Command	Hex value	Decimal value
GTL	01	1
SDC	04	4
GET	08	8
LLO	11	17
DCL	14	20
SPE	18	24
SPD	19	25
LAG	20-3F	32-63
TAG	40-5F	64-95
SCG	60-7F	96-127
UNL	3F	63
UNT	5F	95

Figure C-3
Command codes

Figure C-3 Command codes

Typical command sequences

For the various multiline commands, a specific bus sequence must take place to properly send the command. In particular, the correct listen address must be sent to the instrument before it will respond to addressed commands. Table C-3 lists a typical bus sequence for sending the addressed multiline commands. In this instance, the SDC command is being sent to the instrument. UNL is generally sent as part of the sequence to ensure that no other active listeners are present. Note that ATN is true for both the listen command and the SDC command byte itself.

Table C-3
Typical addressed multiline command sequence

Step	Command	ATN state	Data bus		
			ASCII	Hex	Decimal
1	UNL	Set low	?	3F	63
2	LAG*	Stays low	9	39	57
3	SDC	Stays low	EOT	04	4
4		Returns high			

*Assumes primary address = 25.

Table C-4 gives a typical common command sequence. In this instance, ATN is true while the instrument is being addressed, but it is set high while sending the common command string.

Table C-4
Typical addressed common command sequence

Step	Command	ATN state	Data bus		
			ASCII	Hex	Decimal
1	UNL	Set low	?	3F	63
2	LAG*	Stays low	9	39	57
3	Data	Set high	*	2A	42
4	Data	Stays high	R	52	82
5	Data	Stays high	S	53	83
6	Data	Stays high	T	54	84

*Assumes primary address = 25.

IEEE command groups

Command groups supported by the Model 6482 are listed in Table C-5. Common commands and SCPI commands are not included in this list.

Table C-5
IEEE command groups

HANDSHAKE COMMAND GROUP	
	NDAC=NOT DATA ACCEPTED
	NRFD=NOT READY FOR DATA
	DAV=DATA VALID
UNIVERSAL COMMAND GROUP	
	ATN=ATTENTION
	DCL=DEVICE CLEAR
	IFC=INTERFACE CLEAR
	REN=REMOTE ENABLE
	SPD=SERIAL POLL DISABLE
	SPE=SERIAL POLL ENABLE
ADDRESS COMMAND GROUP	
	LISTEN
	LAG=LISTEN ADDRESS GROUP
	MLA=MY LISTEN ADDRESS
	UNL=UNLISTEN
	TALK
	TAG=TALK ADDRESS GROUP
	MTA=MY TALK ADDRESS
	UNT=UNTALK
	OTA=OTHER TALK ADDRESS
ADDRESSED COMMAND GROUP	
	ACG=ADDRESSED COMMAND GROUP
	GTL=GO TO LOCAL
	SDC=SELECTIVE DEVICE CLEAR
STATUS COMMAND GROUP	
	RQS=REQUEST SERVICE
	SRQ=SERIAL POLL REQUEST
	STB=STATUS BYTE
	EOI=END

Interface function codes

The interface function codes, which are part of the IEEE-488 standards, define an instrument's ability to support various interface functions and should not be confused with programming commands found elsewhere in this manual. The interface function codes for the Model 6482 are listed in Table C-6.

Table C-6
Model 6482 interface function codes

Code	Interface function
SH1	Source Handshake capability.
AH1	Acceptor Handshake capability.
T5	Talker (basic talker, serial poll, unaddressed to talk on LAG).
L4	Listener (basic listener, unaddressed to listen on TAG).
SR1	Service Request capability.
RL1	Remote/Local capability.
PP0	No Parallel Poll capability.
DC1	Device Clear capability.
DT1	Device Trigger capability.
C0	No Controller capability.
E1	Open collector bus drivers.
TE0	No Extended Talker capability.
LE0	No Extended Listener capability.

The codes define Model 6482 capabilities as follows:

SH (Source Handshake Function): SH1 defines the ability of the instrument to initiate the transfer of message/data over the data bus.

AH (Acceptor Handshake Function): AH1 defines the ability of the instrument to guarantee proper reception of message/data transmitted over the data bus.

T (Talker Function): The ability of the instrument to send data over the bus to other devices is provided by the T function. Instrument talker capabilities (T5) exist only after the instrument has been addressed to talk.

L (Listener Function): The ability for the instrument to receive device-dependent data over the bus from other devices is provided by the L function. Listener capabilities (L4) of the instrument exist only after it has been addressed to listen.

SR (Service Request Function): SR1 defines the ability of the instrument to request service from the controller.

RL (Remote-Local Function): RL1 defines the ability of the instrument to be placed in the remote or local modes.

PP (Parallel Poll Function): The instrument does not have parallel polling capabilities (PP0).

DC (Device Clear Function): DC1 defines the ability of the instrument to be cleared (initialized).

DT (Device Trigger Function): DT1 defines the ability of the Model 6482 to have readings triggered.

C (Controller Function): The instrument does not have controller capabilities (C0).

TE (Extended Talker Function): The instrument does not have extended talker capabilities (TE0).

LE (Extended Listener Function): The instrument does not have extended listener capabilities (LE0).

E (Bus Driver Type): The instrument has open-collector bus drivers (E1).

This page left blank intentionally.

Measurement Considerations

In this appendix:

> **NOTE** *This appendix summarizes considerations to make accurate measurements using the Model 6482. For comprehensive information on these and additional measurement considerations, refer to the Low Level Measurements handbook, which is available from Keithley.*

Low current measurements

Low current measurements are subject to a number of error sources that can have a serious impact on measurement accuracy. These aspects are covered in the following paragraphs.

Leakage currents

Leakage currents are generated by high resistance paths between the measurement circuit and nearby voltage sources. These currents can considerably degrade the accuracy of low current measurements.

One way to reduce leakage currents is to use good quality insulators when building the test circuit. Some good quality insulators include Teflon, polyethylene, and sapphire. Avoid materials such as phenolics and nylon.

Humidity may also degrade low current measurements. The amount of water an insulator absorbs will vary depending on the insulator. It is best to choose an insulator on which water vapor does not readily form a continuous film. Sometimes this is unavoidable if the material being measured absorbs water easily, so it is best to make the measurements in an environmentally controlled room. In some cases, an insulator may have ionic contaminants and, especially in high humidity, a spurious current may be generated.

Noise and source impedance

Noise can seriously affect sensitive current measurements. This topic discusses how DUT (device under test) resistance and capacitance affect noise performance.

DUT resistance

The resistance of the DUT will affect the noise performance of the ammeters. As the DUT resistance is reduced, the noise gain of the ammeter will increase. Noise gain can be given by the following equation:

Output V_{NOISE} = Input V_{NOISE} $(1 + R_F/R_{DUT})$

where;

- Output V_{NOISE} is the noise seen at the output of the ammeter.
- Input V_{NOISE} is the noise seen at the input of the ammeter.
- R_F is the internal feedback resistance for the ammeter.
- R_{DUT} is the resistance of the DUT.

Note that as R_{DUT} decreases in value, the output noise increases. For example, when $R_F = R_{DUT}$, the input noise is multiplied by a factor of two. Since decreasing the source resistance can have a detrimental effect on noise performance, there are usually minimum recommended source resistance values based on measurement range. Table D-1 summarizes minimum recommended source resistance values for various measurement ranges for the Model 6482 ammeters. Note

that the recommended source resistance varies by measurement range because the R_F value also depends on the measurement range.

Table D-1
Minimum recommended source resistance values

I-measure range	Minimum recommended source resistance
1 nA – 200 nA 2 µA – 200 µA 2 mA and 20 mA	1 MΩ to 100 MΩ 1 kΩ to 100 kΩ 1 Ω to 100 Ω

Source capacitance

The source capacitance of the device under test (DUT) will also affect the noise performance of the Model 6482 ammeters. In general, as source capacitance increases, the noise gain also increases.

The elements of interest for this discussion are the capacitance (C_{DUT}) of the DUT and the internal feedback capacitance (C_F) for the ammeter. Taking into account the capacitive reactance of these two elements, our previous noise gain formula must be modified as follows:

Output V_{NOISE} = Input V_{NOISE} (1 + Z_F/Z_{DUT})

where;

- Output V_{NOISE} is the noise seen at the output of the ammeter.
- Input V_{NOISE} is the noise seen at the input of the ammeter.
- Z_F is the internal feedback impedance for the ammeter that is formed by C_F and R_F.
- Z_{DUT} is the internal impedance of the DUT that is formed by C_{DUT} and R_{DUT}.

Furthermore,

$$Z_F = \frac{R_F}{\sqrt{(2\pi f R_F C_F)^2 + 1}}$$

and

$$Z_{DUT} = \frac{R_S}{\sqrt{(2\pi f R_S C_S)^2 + 1}}$$

Note that as C_S increases in value, Z_{DUT} decreases in value, thereby increasing the noise gain. Again, at the point where $Z_{DUT} = Z_F$, the input noise is amplified by a factor of two.

Generated currents

Any extraneous generated currents in the test system will add to the configured current, causing errors. Currents can be internally generated, as in the case of instrument input offset current, or they can come from external sources such as insulators and cables.

Offset currents

Internal offset current: The ideal ammeter should read zero when its input terminals are left open. Practical ammeters, however, do have some small current that flows when the input is open. This current is known as the input offset current, and it is caused by bias currents of active devices as well as by leakage currents through insulators within the instrument.

The internal input offset current adds to the source current so that the meter measures the sum of the two currents:

$$I_M = I_S + I_{I0}$$

Where: I_M is the measured current.
      I_S is the source current.
      I_{I0} is the internal input offset current.

External offset current: Offset currents can also be generated from external effects, such as electrochemical effect (discussed below). The external offset current also adds to the source current, and the ammeter again measures the sum of the currents:

$$I_M = I_S + I_{I0} + I_{EO}$$

where: I_{EO} is the external offset current.

As long as the internal and external offsets remain stable for a reasonable period of time, the relative feature of the Model 6482 can be used to zero out offset current. With the source current (I_S) set to zero, the ammeter will only measure and display the offset current reading. When REL is enabled, the display will zero. What happens is that the offset current reading is acquired as the rel value and is subtracted from present and future readings. When the source current is applied, the displayed reading will not include the offset:

$$I_M = I_S + I_{I0} + I_{EO} - rel$$
$$I_M = I_S$$

where: rel is the rel value ($I_{I0} + I_{EO}$).

When the rel value equals the sum of the offsets, only the source current will be displayed. See Section 5 for details on relative.

Offset current drift is a function of time and/or temperature. Therefore, when offset current drifts significantly, you will have to re-zero the ammeter.

Electrochemical effects

Error currents also arise from electrochemical effects when ionic chemicals create weak batteries on a circuit board. These batteries could generate a few nanoamps of current between conductors. Ionic contamination may be the result of body oils, salts, or solder flux. The problem is further enhanced by high humidity (moisture) that decreases insulation resistance.

When building test fixtures, select insulators that resist water absorption, and use the fixture in a moderate humidity environment. Also, be sure that all insulators are kept clean and free of contamination.

Humidity

Excess humidity can reduce insulation resistance on PC boards and in test connection insulators. Reduction in insulation resistance can, of course, seriously affect high-
impedance measurements. Also, humidity (moisture) can combine with contaminants to produce offset currents caused by electrochemical effects (see "Electrochemical effects"). To minimize the effects of moisture, keep humidity to a minimum (ideally <50 percent), and keep components and connectors in the test system clean.

Triboelectric effects

Triboelectric currents are generated by charges created between a conductor and an insulator due to friction. Here, free electrons rub off the conductor and create a charge imbalance that causes the current flow. For example, bending a triaxial cable causes friction between the center conductor (HI) and its surrounding insulator resulting in triboelectric currents.

Triboelectric currents can be minimized as follows:

- Use low noise cables. These cables are specially designed to minimize charge generation and use graphite to reduce friction.
- Use the shortest cables possible, and secure them (for example, taping or tying) to a non-vibrating surface to keep them from moving.

Piezoelectric and stored charge effects

Piezoelectric currents are generated when mechanical stress is applied to certain insulating materials (for example, crystalline). In some plastics, pockets of stored charge cause the material to behave in a similar manner.

When building test fixtures, choose good insulating materials and make connecting structures as rigid as possible. Make sure there are no mechanical stresses on the insulators.

Dielectric absorption

Dielectric absorption in an insulator can occur when a voltage across that insulator causes positive and negative charges within the insulator to polarize because various polar molecules relax at different rates. When the voltage is removed, the separated charges generate a decaying current through circuits connected to the insulator as they recombine.

To minimize the effects of dielectric absorption on current measurements, avoid applying voltages greater than a few volts to insulators being used for sensitive current measurements. In cases where this practice is unavoidable, it may take minutes or even hours in some cases for the current caused by dielectric absorption to dissipate.

Voltage burden

The input resistance of the ammeter causes a small voltage drop across the input terminals. This voltage is known as the voltage burden. If the voltage burden is large in relation to the voltage of the measured circuit, then significant measurement errors will occur.

Refer to Figure D-1 to see how voltage burden affects current measurements. Assume V_S is set to output 5mV and R_L is 5 kΩ. An ideal ammeter with zero voltage burden would measure the current source as follows:

$$I_M = V_S/R_S$$
$$= 5 \text{ mV}/5 \text{ k}\Omega$$
$$= 1$$

In practice however, every ammeter has a voltage burden. If the voltage burden (V_B) is 1 mV, the current will be measured as follows:

$$I_M = (V_S + V_B)/R_S$$
$$= (5 \text{ mV} + 1 \text{ mV})/5 \text{ k}\Omega$$
$$= 1.2$$

The 1 mV voltage burden caused a 20% measurement error.

Figure D-1
Voltage burden

$$I_M = \frac{V_S - V_B}{R_L}$$
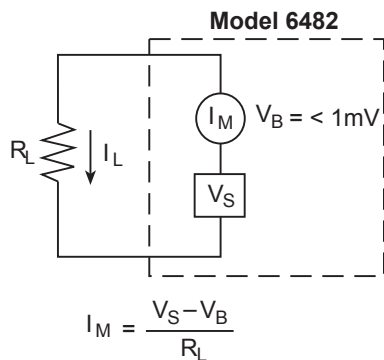
General measurement considerations

The following measurement considerations apply to all precision measurements.

Ground loops

Ground loops that occur in multiple-instrument test setups can create error signals that cause erratic or erroneous measurements. The configuration shown in Figure D-2 introduces errors in two ways. Large ground currents flowing in one of the wires will encounter small resistances, either in the wires, or at the connecting points. This small resistance results in voltage drops that can affect the measurement. Even if the ground loop currents are small, magnetic flux cutting across the large loops formed by the ground leads can induce sufficient voltages to disturb sensitive measurements.

To prevent ground loops, instruments should be connected to ground at only a single point, as shown in Figure D-3. Note that only a single instrument is connected directly to power line ground. Experimentation is the best way to determine an acceptable arrangement. For this purpose, measuring instruments should be placed on their lowest ranges. The configuration that results in the lowest noise signal is the one that should be used.

Figure D-2
Power line ground loops
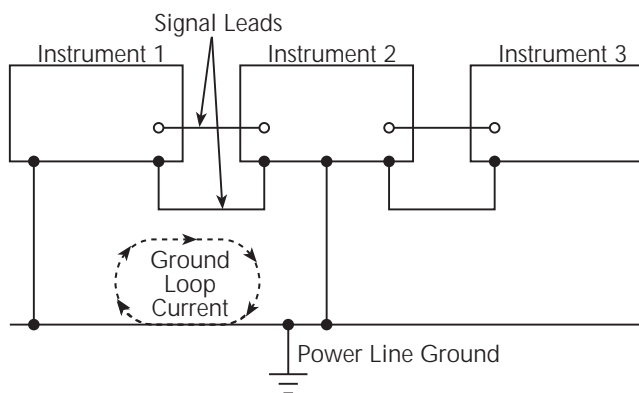
Figure D-3
Eliminating ground loops

Instrument 1   Instrument 2   Instrument 3

Power Line Ground
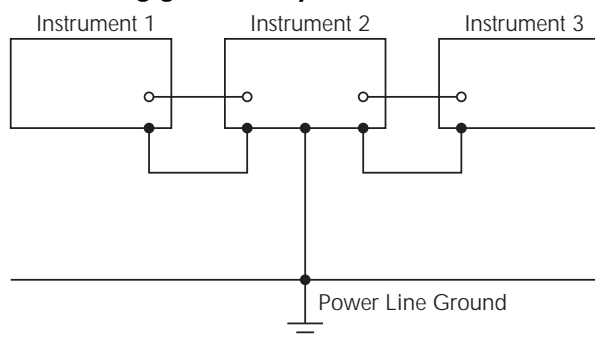
Light

Some components, such as semiconductor junctions and MOS capacitors on semiconductor wafers, are excellent light detectors. Consequently, these components must be tested in a light-free environment. While many test fixtures provide adequate light protection, others may allow sufficient light penetration to affect the test results. Areas to check for light leaks include doors and door hinges, tubing entry points, and connectors or connector panels.

Electrostatic interference

Electrostatic interference occurs when an electrically charged object is brought near an uncharged object, thus inducing a charge on the previously uncharged object. Usually, effects of such electrostatic action are not noticeable because low impedance levels allow the induced charge to dissipate quickly. However, the high impedance levels of many measurements do not allow these charges to decay rapidly, and erroneous or unstable readings may result. These erroneous or unstable readings may be caused in the following ways:

- DC electrostatic field can cause undetected errors or noise in the reading.
- AC electrostatic fields can cause errors by driving the input preamplifier into saturation, or through rectification that produces DC errors.

Electrostatic interference is first recognizable when hand or body movements near the experiment cause fluctuations in the reading. Means of minimizing electrostatic interference include:

1. Shielding. Possibilities include: a shielded room, a shielded booth, shielding the sensitive circuit, and using shielded cable. The shield should always be connected to a solid connector that is connected to signal low. If circuit low is floated above ground, observe safety precautions, and avoid touching the shield. Meshed screen or loosely braided cable could be inadequate for high impedances, or in strong fields. Note, however, that shielding can increase capacitance in the measuring circuit, possibly slowing down response time.
2. Reduction of electrostatic fields. Moving power lines or other sources away from the   experiment reduces the amount of electrostatic interference seen in the measurement.

Magnetic fields

A magnetic field passing through a loop in a test circuit will generate a magnetic EMF (voltage) that is proportional to the strength of the field, the loop area, and the rate at which these factors are changing. Magnetic fields can be minimized by following these guidelines:

- Locate the test circuit as far away as possible from such magnetic field sources as motors, transformers and magnets.
- Avoid moving any part of the test circuit within the magnetic field.
- Minimize the loop area by keeping leads as short as possible and twisting them together.

Electromagnetic interference (EMI)

The electromagnetic interference characteristics of the Model 6482 comply with the electromagnetic compatibility (EMC) requirements of the European Union as denoted by the CE mark. However, it is still possible for sensitive measurements to be affected by external sources. In these instances, special precautions may be required in the measurement setup.

Sources of EMI include:

- Radio and TV broadcast transmitters.
- Communications transmitters, including cellular phones and handheld radios.
- Devices incorporating microprocessors and high-speed digital circuits.
- Impulse sources as in the case of arcing in high-voltage environments.

The effect on instrument performance can be considerable if enough of the unwanted signal is present. The effects of EMI can be seen as an unusually large offset, or, in the case of impulse sources, erratic variations in the displayed reading.

The instrument and experiment should be kept as far away as possible from any EMI sources. Additional shielding of the instrument, experiment and test leads will often reduce EMI to an acceptable level. In extreme cases, a specially constructed screen room may be required to sufficiently attenuate the troublesome signal.

External filtering of the input signal path may be required. In some cases, a simple one-pole filter may be sufficient. In more difficult situations, multiple notch or band-stop filters, tuned to the offending frequency range, may be required. Connecting multiple capacitors of widely different values in parallel will maintain a low impedance across a wide frequency range. Keep in mind, however, that such filtering may have detrimental effects (such as increased response time) on the measurement.

In this appendix:

Introduction

The Model 6482 supports two GPIB protocols: SCPI and 488.1. The 488.1 protocol is included to significantly increase speed over the GPIB.

When using the 488.1 protocol, throughput is enhanced up to 10 times for data sent to the Model 6482 (command messages) and up to 20 times for data returned by the Model 6482 (response messages). The speed of readings sent over the GPIB is also increased.

> *NOTE*   *With the 488.1 protocol selected, you will still use SCPI commands to program the* Model 6482. *Operation differences between the two protocols are discussed in this appendix.*

Selecting the 488.1 protocol

Perform the following steps to select the 488.1 protocol:

1. Press the **MENU** key to display the MAIN MENU.
2. Place the cursor on **COMMUNICATION**, and press the **ENTER** key to display the COMMUNICATIONS SETUP menu.
3. Place the cursor on **GPIB**, and press the **ENTER** key to display the present GPIB address.
4. Press the **ENTER** key to display the GPIB PROTOCOL menu.
5. Place the cursor on **488.1**, and then press the **ENTER** key.
6. Use the **EXIT** key to return to the previous menu.

When switching between the SCPI protocol and 488.1 protocol, the instrument does not reset. The GPIB protocol setting is saved in EEPROM, and the instrument will power up with that selected protocol.

The GPIB protocol cannot be changed over the bus. However, there is a query command to determine the presently selected protocol. When the 488.1 protocol is selected, the message exchange protocol (MEP) disables. Therefore, if you use the following query to request the state of MEP, you will know which protocol is enabled:

```
:SYSTem:MEP[:STATe]?
```

If a 1 is returned, MEP is enabled, and the SCPI protocol is selected. A 0 indicates that MEP is disabled, and the 488.1 protocol is enabled. To summarize:

1 = SCPI protocol

0 = 488.1 protocol

Protocol differences

The following information covers the differences between the 488.1 protocol and the SCPI protocol.

Message exchange protocol (MEP)

When the 488.1 protocol is selected, the MEP is disabled to speed up GPIB operation.

The following guidelines/limitations must be followed when using the 488.1 protocol:

*   If a query is sent, it must be the only command on the line (this limitation also means no multiple queries can be sent). Otherwise, full SCPI command syntax is still supported including long-form and short form commands, multiple commands, and MIN/MAX/DEF parameter definitions.

For example, the following command strings are invalid:

```
:SOUR2:VOLT:RANG 10;*OPC?
:SENS2:CURR:RANG?;::READ?
:READ?;:READ?
```

The following command strings are valid:

```
:SOUR1:VOLTage:STARt 1;STOP 10;step 1
:curr:nplc 1.0;::curr:rang min
:SENS2:CURR:RANG? MAX
:READ?
```

- When a query is sent, either the data must be read back or a Device Clear (DCL) or Interface Clear (IFC) must be performed to reset the query.
- When sending a command or query, do not attempt to read data from the Model 6482 until the terminator has been sent (usually Line Feed with EOI). Otherwise, a DCL or IFC must be sent to reset the input parser.
- When receiving data, all data, up to and including the terminator (LF with EOI), must be accepted. Otherwise, a DCL or IFC must be sent to reset the output task.
- Empty command strings (terminator only) should not be sent.

Using SCPI-based programs

In general, an existing SCPI-based program will run properly and faster in the 488.1 protocol if it meets the above guidelines and limitations.

Bus hold-off

*OPC, *OPC?, and *WAI are still functional but are not needed for the 488.1 protocol. When sending commands, the GPIB is automatically held off when it detects a terminator. The hold-off is released when all the commands have finished executing, or if there is some parser or command error. An exception is an initiate command, which releases the hold-off immediately and does not wait for all of the readings to be acquired. This immediate release of bus hold-off is done to support GET, SDC, IFC, *TRG, *RCL, *RST, SYSTem:PRESet and ABORt during data acquisition.

Trigger-on-talk

Trigger-on-talk functionality has been added for the 488.1 protocol. If a query has not been received by the instrument, the Model 6482 will automatically assume a READ? command has been sent when it is addressed to talk. This technique increases GPIB speed by decreasing the transmission and parser times for the command.

Trigger-on-talk is extremely useful in the single-shot reading mode (*RST default) and is the main reason for a >2x speed improvement over the SCPI protocol. Remember that the output must be on (:OUTPut[1]:STATe ON or :OUTPut2:STATe ON) before you can take readings.

The ARM:SOUR BUS and ARM:COUN INF commands are not supported by READ? with the 488.1 protocol selected. If you send one of these commands, a DCL or IFC may be required to reset the GPIB.

Message available

The MAV (message available) bit in the Serial Poll byte will be set when the query is finished being processed, not when there is data available in the output buffer (as with the SCPI protocol). For the 488.1 protocol, output data will not be formatted until the first request for data is received. This delay may cause unexpected time-outs when using SRQ on MAV for queries that take a long time to execute.

General operation notes

- The TALK, LSTN, and SRQ indicators are not functional in the 488.1 protocol. This speeds up data throughput greatly. The REM indicator still operates since it is critical to fundamental GPIB operation.
- If the instrument is in REMote, the GTL command may not put the Model 6482 into the local mode. Only the front panel LOCAL key is guaranteed to operate, if not in local lockout (LLO). GTL will still disable LLO.
- IEEE-488 bus commands and features (GET, IFC, SDC, DCL, LLO, Serial Poll, and SRQ) are still fully supported.
- Multiple TALKs on the same query are supported as in the SCPI protocol. This feature is useful when reading back long ASCII strings.

Example Programs

In this appendix:

Introduction

This section provides listings of pseudo-code for example programs throughout this manual. These programs include:

- Basic measurement program from Section 3.
- Data store program from Section 6.
- Sweep program from Section 7.
- Limit test program from Section 10.

Basic measurement program

The program listing below performs basic single-channel device under test (DUT) measurement as covered in Section 3. This program sets up the following operating modes:

- Channel 2 measurement range: 2 µA
- Channel 2 source range: 10 V
- Channel 2 source output level: 10 V

```
'Basic Measurement Program
*RST                                    ' Restore GPIB Defaults
:SENS2:CURR:RANG 2E-6                    ' Select 2 uA Channel 2 measure range
:FORM:ELEM CURR2                         ' Select Channel 2 reading.

:SOUR2:VOLT:RANG 10                      ' Select 10 V Channel 2 source range.
:SOUR2:VOLT 10                           ' Output 10 V on channel 2.
:OUTP2 ON                                ' Turn on channel 2 output.
:READ?                                   ' Trigger and request reading.

' Read back result

:OUTP2 OFF                               ' Turn off channel 2 output.
```

Data store program

The program listing below performs data storage as covered in Section 6, and it also demonstrates how to use SRQs to determine when storage is complete. This program sets up the following operating modes:

- Number of points: 10
- Trigger count: 10
- Data store source: channel 1 readings only
- Acquired data: all buffer readings, mean (average), and standard deviation.

```
' Data Store Program
*RST                                    ' Restore GPIB defaults.
:STAT:MEAS:ENAB 512;*SRE 1              ' Enable buffer full SRQ.
:SOUR1:VOLT 10                          ' Output 10 V on channel 1.
:SOUR1:DEL 0.5                          ' 0.5 sec source 1 delay.
:TRAC:POIN 10                           ' Store 10 readings.
:TRAC:FEED:CONT NEXT                    ' Turn on data store.
:TRIG:COUN 10                           ' Trigger count = data store points.
:FORM:ELEM:TRAC CURR1                   ' Select channel 1 current.
:OUTP1 ON                               ' Turn on channel 1 output.
:INIT                                   ' Trigger readings.

' Wait for buffer to fill

:OUTP1 OFF                              ' Turn off channel 1 output.
:TRAC:DATA?                             ' Request all data store readings.

' Read back result

:DISP:MODE CALC3                        ' Select channel 1 for stats.
:CALC8:FORM MEAN                        ' Select mean buffer statistic.
:CALC8:DATA?                            ' Request mean reading.

' Read back result

:CALC8:FORM SDEV                        ' Standard deviation statistic.
:CALC8:DATA?                            ' Request standard deviation.

' Read back result
```

Linear sweep program

The program listing below performs a linear sweep as covered in Section 7. This program sets up the following operating modes:

- Source and measure channel: channel 2
- Source mode: sweep
- Current range: 20 mA
- Start voltage: 1 V
- Stop voltage: 10 V
- Step voltage: 1 V
- Source delay: 100 ms

```
' Linear Sweep Program
*RST                                     ' Restore GPIB defaults.
:FORM:ELEM CURR2                         ' Select channel 2 measure data.
:SENS2:CURR:RANG 20e-3                   ' Enable 20 mA measure range.
:SOUR2:VOLT:START 1                      ' 1 V Channel 2 start voltage.
:SOUR2:VOLT:STOP 10                      ' 10 V Channel 2 stop voltage.
:SOUR2:VOLT:STEP 1                       ' 1 V Channel 2 step voltage.
:SOUR2:VOLT:MODE SWE                     ' Select Channel 2 source sweep mode.
:SOUR2:SWE:RANG AUTO                     ' Channel 2 automatic source range mode.
:SOUR2:SWE:SPAC LIN                      ' Channel 2 linear sweep mode.
:TRIG:COUN 10                           ' Trigger count = sweep points.
:SOUR2:DEL 0.1                          ' Source delay = 0.1 s.
:OUTP2 ON                               ' Turn on channel 2 output.
:READ?                                  ' Trigger and acquire readings.

' Read back result

:OUTP2 OFF                              ' Turn off channel 2 output.
```

Limit test program

The program listing below performs a limits test as covered in Section 10. This program sets up the following operating modes:

- Source and measure channel: channel 1
- Source voltage: 1 V
- Source delay: 100 ms
- Limit 3 upper value: 10 mA
- Limit 3 lower value: 4 mA
- Limit 4 upper value: 8 mA
- Limit 4 lower value: 6 mA

```
' Limit Test Program
*RST                                    ' Restore GPIB defaults.
:SENS1:CURR:RANG:AUTO ON                ' Channel 1 autorange.
:SOUR1:VOLT:TRIG 10                     ' Source 1 V when triggered.
:SOUR1:DEL 0.1                          ' 0.1 s source delay.

:CALC7:FEED CALC3                       ' Use Channel 1 data for limit tests.
:CALC7:LIM3:UPP 10e-3                   ' Limit 3 upper limit = 10 mA.
:CALC7:LIM3:LOW 4e-3                    ' Limit 3 lower limit = 4 mA.
:CALC7:LIM4:UPP 8e-3                    ' Limit 4 upper limit = 8 mA.
:CALC7:LIM4:LOW 6e-3                    ' Limit 4 lower limit = 6 mA.
:CALC7:LIM1:STAT 0                      ' Turn off Limit 1.
:CALC7:LIM3:STAT 1                      ' Turn on Limit 3.
:CALC7:LIM4:STAT 1                      ' Turn on Limit 4.
:OUTP1 ON                              ' Turn on channel 1 output.
:INIT                                  ' Trigger readings.
:OUTP1 OFF                             ' Turn off channel 1 output.
:CALC7:LIM3:FAIL?                       ' Request Limit 3 fail status.

' Read back result

:CALC7:LIM4:FAIL?                       ' Request Limit 4 fail status.

' Read back result
```

This page left blank intentionally.

Index

This page left blank intentionally.

Specifications are subject to change without notice.
All Keithley trademarks and trade names are the property of Keithley Instruments, Inc.
All other trademarks and trade names are the property of their respective companies.

KEITHLEY

A GREATER MEASURE OF CONFIDENCE