

# **Agilent EasyEXPERT**

## **User's Guide**

Volume 1



**Agilent Technologies**

# Notices

© Agilent Technologies, Inc. 2013

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

## Manual Part Number

B1540-90000

## Edition

Edition 1, June 2013

Agilent Technologies, Inc.  
5301 Stevens Creek Blvd  
Santa Clara, CA 95051 USA

## Warranty

**The material contained in this document is provided “as is,” and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.**

## Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

## Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as “Commercial computer software” as defined in DFAR 252.227-7014 (June 1995), or as a “commercial item” as defined in FAR 2.101(a) or as “Restricted computer software” as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies’ standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will

receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

---

## For B1500A Users

Agilent B1500A supports the following measurement resources. For reading this manual, ignore the information about the other resources.

- HPSMU High power source/monitor unit (B1510A)
- MPSMU Medium power source/monitor unit (B1511A/B1511B)
- MCSMU Medium current source/monitor unit (B1514A)
- HRSMU High resolution source/monitor unit (B1517A)
- MFCMU or CMU Multi frequency capacitance measurement unit (B1520A)
- HVSPGU or SPGU High voltage semiconductor pulse generator unit (B1525A)
- WGFMU Waveform generator/fast measurement unit (B1530A)

---

## For B1505A Users

Agilent B1505A supports the following measurement resources. For reading this manual, ignore the information about the other resources.

- HPSMU High power source/monitor unit (B1510A)
- MPSMU Medium power source/monitor unit (B1511A)
- HCSMU High current source/monitor unit (B1512A)
- MCSMU Medium current source/monitor unit (B1514A)
- HVSMU High voltage source/monitor unit (B1513A/B1513B)
- MFCMU or CMU Multi frequency capacitance measurement unit (B1520A)
- HVMCU High voltage medium current unit (N1266A with one B1513B and two B1514A/B1512A)
- UHCU Ultra high current unit (N1265A with two B1514A/B1512A)
- UHVU Ultra high voltage unit (N1268A with two B1514A/B1512A)

---

## In This Manual

This manual describes the reference information of Agilent Technologies EasyEXPERT which is the system software of Agilent Technologies B1500 device analyzer series. This manual consists of the following chapters.

1. Main GUI  
Provides the reference information of the EasyEXPERT main GUI.
2. Classic Test Definition  
Provides the reference information of the classic test setup GUI.
3. Application Test Definition  
Provides the reference information of the application test definition setup GUI.
4. Function Details  
Explains the several functions and the initial settings of the B1500/EasyEXPERT.
5. Built-in Programming Tool  
Provides the reference information of the built-in mathematical functions and the read-out functions.

User's Guide Vol. 2 consists of the following chapters.

6. Remote Control Interface
7. Utilities
8. Application Library
9. If You Have a Problem
10. Error Messages

---

### NOTE

To get the latest firmware/software/manual/support information, go to [www.agilent.com](http://www.agilent.com) and type in EasyEXPERT in the Search field at the top of the page.

---

---

# Contents

## 1. Main GUI

Start EasyEXPERT .....	1-4
Workspace Configurator .....	1-5
If there is no workspace.....	1-5
If there is one workspace.....	1-6
If there are multiple workspace.....	1-8
Main Screen .....	1-11
Main Screen GUI.....	1-12
Run Option .....	1-19
Data Display Manager .....	1-20
Data Display Properties .....	1-21
Test Result Editor .....	1-23
Test Results Data Filter .....	1-23
Export in My Format.....	1-24
Test Results Data Folder Export.....	1-24
Test Results Data Auto Export.....	1-26
Test Results Data Properties .....	1-28
Application Test .....	1-29
Tracer Test.....	1-31
Graph Setup Tool.....	1-37
Option Tool .....	1-37
Replay traces.....	1-39
Select reference traces .....	1-40
Select tracking traces.....	1-41
Color and thickness .....	1-41
Oscilloscope View.....	1-42
Arithmetic Operation Area .....	1-47
Quick Test .....	1-48
Repeat Measurement Setup .....	1-50

---

# Contents

Thermo-trigger List .....	1-53
Organize Preset Group .....	1-56
Calibration .....	1-58
Module Self Calibration .....	1-58
SMU Zero Cancel .....	1-59
CMU Calibration .....	1-60
Configuration .....	1-64
Main Frame .....	1-64
Modules .....	1-65
ASU .....	1-66
Switching Matrix .....	1-67
SMU/PG Selector .....	1-69
Module Selector .....	1-69
Dual HCSPMU Combination .....	1-71
UHC Expander / Fixture .....	1-72
HVSPMU Current Expander .....	1-75
UHV Expander .....	1-77
SMU Output Setting Limits .....	1-78
Event Log .....	1-78
Extended Configuration .....	1-79
Switching Matrix Operation Panel .....	1-81
Standby Channel Definition .....	1-83
Data Display .....	1-84
Data Display GUI .....	1-86
Display Setup .....	1-95
Graph Properties .....	1-96
List Display Properties .....	1-97
Tool Bar .....	1-97
Data Status .....	1-100

---

# Contents

Setup Summary . . . . .	1-101
Miscellaneous Operations . . . . .	1-102
Preview . . . . .	1-108
Preview Window GUI . . . . .	1-108
Preview Settings . . . . .	1-110
Text File Export settings . . . . .	1-111

## 2. Classic Test Definition

I/V Sweep . . . . .	2-4
Channel Setup . . . . .	2-5
Measurement Setup . . . . .	2-6
Multi Channel I/V Sweep . . . . .	2-9
Channel Setup . . . . .	2-10
Measurement Setup . . . . .	2-11
I/V List Sweep . . . . .	2-13
Channel Setup . . . . .	2-14
Measurement Setup . . . . .	2-15
Define vector data . . . . .	2-18
I/V-t Sampling . . . . .	2-21
Channel Setup . . . . .	2-22
Measurement Setup . . . . .	2-23
C-V Sweep . . . . .	2-26
Channel Setup . . . . .	2-27
Measurement Setup . . . . .	2-28
Direct Control . . . . .	2-30
Channel Setup . . . . .	2-31
Measurement Setup . . . . .	2-33
Advanced Setup . . . . .	2-34
Command Setup . . . . .	2-36

---

# Contents

compenReZ .....	2-38
compenImZ .....	2-39
compenReY .....	2-40
compenImY .....	2-41
Function Setup .....	2-42
Auto Analysis Setup .....	2-43
Display Setup .....	2-46
SMU Range Setup Window .....	2-48
ADC and Integration Time Setup Window .....	2-50
Pulse Setup Window .....	2-51
Advanced Setup Window .....	2-53
Range Setup Window .....	2-55
Advanced Setup Window for C-V Sweep .....	2-57
Switching Matrix Control .....	2-59
SPGU Control .....	2-61
Advanced Setup Window for SPGU Control .....	2-62
SPGU Pulse Setup Window .....	2-63
Load Z Setup Window .....	2-65
Pulse Switch Setup Window .....	2-66
SPGU ALWG Setup Window .....	2-67
Define ALWG Waveform Window .....	2-68
Waveform Pattern Editor .....	2-71
<b>3. Application Test Definition</b>	
Test Definition Window .....	3-3



---

# Contents

Test Specification .....	3-5
Define Layout .....	3-10
Test Output .....	3-14
Analysis Parameter Definition .....	3-15
Test Contents .....	3-16
Defining/Editing Test Contents .....	3-17
Debugging Test Contents .....	3-21
Variable Inspector .....	3-22
External Variable Setup .....	3-23
Local Variable Definition .....	3-24
Program Component .....	3-26
Auto Analysis .....	3-27
Data Display Control .....	3-30
Display Data Setup .....	3-31
GPIB I/O .....	3-33
Message .....	3-38
Data Store Control .....	3-39
Command Execution .....	3-40
Using Command Execution .....	3-42
Command Parameters .....	3-42
Defining Numeric/Vector Input Parameter .....	3-43
Defining String/Numeric Input Parameters .....	3-43
Defining Numeric Output Parameter .....	3-44
Defining Vector Output Parameter .....	3-44
Defining String Output Parameter .....	3-44
Defining Format Field .....	3-45
Setup Example .....	3-46

---

# Contents

## 4. Function Details

I/V Sweep Measurement . . . . .	4-3
Available Measurement Resources . . . . .	4-3
Output Range . . . . .	4-4
Basic Sweep Measurement . . . . .	4-4
Subordinate Sweep Measurement . . . . .	4-6
Synchronous Sweep Measurement . . . . .	4-8
Pulsed Sweep Measurement . . . . .	4-9
Multi Channel I/V Sweep Measurement . . . . .	4-12
I/V-t Sampling Measurement . . . . .	4-13
Available Measurement Resources . . . . .	4-13
Operation Summary . . . . .	4-13
Setup Parameters . . . . .	4-15
Time Data and Index . . . . .	4-17
Source Output Sequence and Time Origin . . . . .	4-18
Sampling Completion . . . . .	4-18
C-V Sweep Measurement . . . . .	4-19
Available Measurement Resources . . . . .	4-19
Setup Parameters . . . . .	4-19
Measurement Parameters . . . . .	4-22
Four-Terminal Pair Configuration . . . . .	4-24
Error Correction . . . . .	4-25
SPGU Module . . . . .	4-27
PG Operation Mode . . . . .	4-28
ALWG Operation Mode . . . . .	4-35
Sweep Abort Function . . . . .	4-37
When abort occurs . . . . .	4-37
Standby Function . . . . .	4-38
Standby Channels . . . . .	4-38

---

# Contents

Standby State . . . . .	4-38
Output Sequence of Standby Channels . . . . .	4-39
To Use Standby Function . . . . .	4-40
Bias Hold Function . . . . .	4-41
Current Offset Cancel . . . . .	4-43
SMU CMU Unify Unit . . . . .	4-45
Atto Sense and Switch Unit . . . . .	4-47
SMU/PG Selector . . . . .	4-49
Module Selector . . . . .	4-50
Ultra High Current Expander/Fixture . . . . .	4-51
HVSMU Current Expander . . . . .	4-53
Ultra High Voltage Expander . . . . .	4-54
SMU Ranging Mode . . . . .	4-55
To Set Ranging Mode . . . . .	4-55
Auto Ranging . . . . .	4-56
Limited Auto Ranging . . . . .	4-57
Fixed Range . . . . .	4-57
Compliance Range . . . . .	4-57
Enhanced Auto Ranging for Current Measurement . . . . .	4-58
SMU Compliance . . . . .	4-59
Polarity and Output Area . . . . .	4-59
Power Compliance . . . . .	4-60
To Set Compliance . . . . .	4-61
SMU Pulse . . . . .	4-62
To Set Pulse Output . . . . .	4-62
Pulse Setup Parameters . . . . .	4-63
SMU Measurement Time . . . . .	4-65

---

# Contents

Integration Time . . . . .	4-65
Overhead Time . . . . .	4-67
To Specify Source Output Time . . . . .	4-67
Wait Time . . . . .	4-68
Multiple Measurement Channels . . . . .	4-69
SMU Filter . . . . .	4-70
SMU Series Resistor . . . . .	4-71
Interlock Function . . . . .	4-72
Auto Power Off Function . . . . .	4-73
Initial Settings . . . . .	4-74
<b>5. Built-in Programming Tool</b>	
Variables and Expressions . . . . .	5-3
Expressions . . . . .	5-4
Built-in Functions . . . . .	5-7
Read-out Functions . . . . .	5-16
Functions for Marker . . . . .	5-16
Functions for Cursor . . . . .	5-17
Functions for Line . . . . .	5-17
Functions for Two Line Display . . . . .	5-18
Script Program Statements . . . . .	5-20

---

**1**

**Main GUI**

---

## Main GUI

This chapter provides the reference information of Agilent EasyEXPERT main GUI.

- “Start EasyEXPERT”
- “Workspace Configurator”
- “Main Screen”
- “Application Test”
- “Tracer Test”
- “Quick Test”
- “Repeat Measurement Setup”
- “Organize Preset Group”
- “Calibration”
- “Configuration”
- “Switching Matrix Operation Panel”
- “Standby Channel Definition”
- “Data Display”
- “Preview”
- “Text File Export settings”

---

### NOTE

#### Touch Screen Operation

If you use touch screen operation than mouse, change the font size of menu. Font size 14 is recommended for the touch screen operation.

Open the Control Panel, select *Appearance and Themes*, and select *Display* to open the Display Properties dialog box. Select the *Appearance* tab, and click the *Advanced* button to open the Advanced Appearance dialog box. On the dialog box, set *Item* to *Menu*, and set the *Size of Font* to 14.

---

To start EasyEXPERT, see “Start EasyEXPERT” on page 1-4 and “Workspace Configurator” on page 1-5.

To perform application test setup, see “Application Test” on page 1-29.

To perform classic test setup, see “Classic Test Definition” on page 2-1.

To perform tracer test setup, see “Tracer Test” on page 1-31.

To perform quick test setup, see “Quick Test” on page 1-48.

To perform measurement, see “Main Screen” on page 1-11.

To display measurement data and perform data analysis, see “Data Display” on page 1-84.

To perform self-test or diagnosis, see “Configuration” on page 1-64.

To perform self-calibration, see “Calibration” on page 1-58.

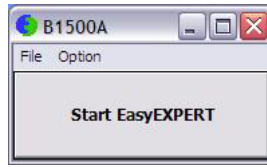
To control Agilent B2200A/B2201A/E5250A switching matrix connection, see “Switching Matrix Operation Panel” on page 1-81.

To set the standby channels and the standby mode, see “Standby Channel Definition” on page 1-83.

To define your application test, see “Application Test Definition” on page 3-1.

## Start EasyEXPERT

The Start EasyEXPERT window is opened after the B1500 is booted up or by using the Start EasyEXPERT menu in the Start menu or the Start EasyEXPERT icon on Windows desktop. This window is not opened after the boot if the automatic start function is effective. This window provides the following graphical user interface.



**Start EasyEXPERT** This button launches EasyEXPERT.

**File menu** Provides the following function.

**Exit** Closes the Start EasyEXPERT window.

**Option menu** Provides the following functions.

**Auto Start of EasyEXPERT** Enables or disables the automatic start function. If this label is checked, the function is enabled. From the next boot or logon, the Start EasyEXPERT window will be skipped and EasyEXPERT will be launched. To disable this function, use the Start EasyEXPERT window opened by the File > Exit menu on the EasyEXPERT main screen. This function is not effective for Desktop EasyEXPERT.

**EasyEXPERT Database** Provides the following functions.

- Backup EasyEXPERT Database  
Opens the EasyEXPERT Database Backup Creation Wizard used to make backup of the EasyEXPERT database. You can make backup easily by following the wizard. Required time depends on the database size.
- Restore EasyEXPERT Database  
Opens the EasyEXPERT Database Restoration Wizard used to restore the EasyEXPERT database backup. You can restore the backup easily by following the wizard. Required time depends on the database size.

EasyEXPERT Database is the folder used to store your test setup data and such.



## Workspace Configurator

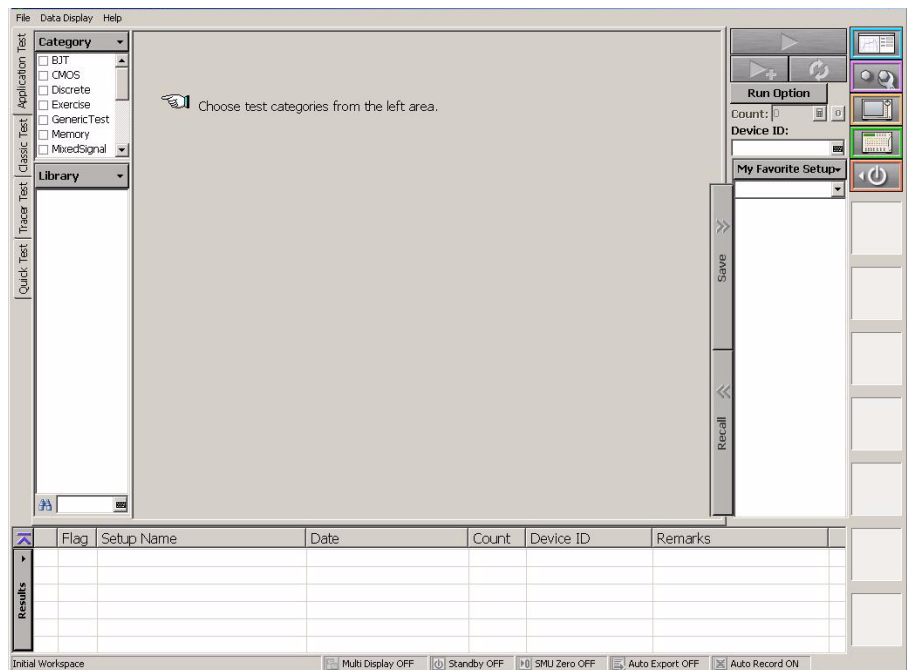
Workspace means the space created in the B1500 internal hard disk drive, and is used to store the test setup, test result data, and so on. The workspace can be created and allocated for each user.

The workspace configurator will run after you launch Agilent EasyEXPERT. The workspace configurator displays a screen used to select the workspace you use. There are three types of display, depending on the number of existing Workspace.

- “If there is no workspace”
- “If there is one workspace”
- “If there are multiple workspace”

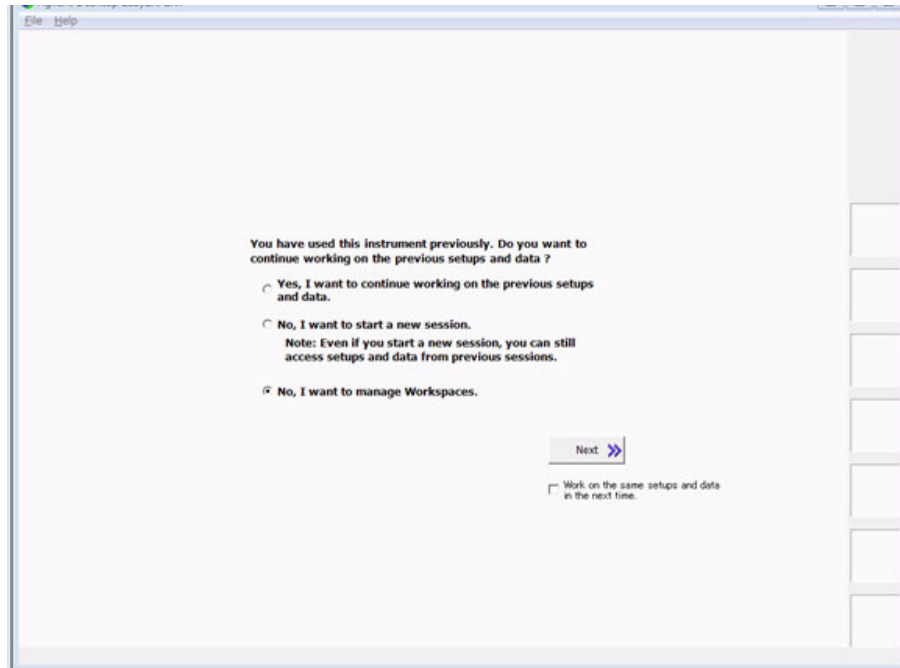
### If there is no workspace

EasyEXPERT will automatically create a workspace (Name: Initial Workspace, Scope: Private, Owner: Windows login account) and display the EasyEXPERT main screen.



## If there is one workspace

You will see the screen asking you if you want to create a workspace or not. Select Yes or No for the message and click OK.



- **OK**  
This button displays the EasyEXPERT main screen using the specified workspace.
- *Work on the same setups and data in the next time.*  
If this check box is checked, the workspace selection screen will be skipped at the next startup and EasyEXPERT will be launched with the workspace used at the last operation. To perform this setup again, click the File > Close Workspace menu on the EasyEXPERT main screen.
- *Yes, I want to continue working on the previous setups and data.*  
This button must be selected if you do not want to create a new workspace, and want to use the workspace previously used.  
The OK button will work to open the EasyEXPERT main screen.

- *No, I want to start a new session.*

This button must be selected if you want to create a new workspace, and want to use it instead of the workspace previously used. If you select this button, the label of the OK button will change to Next.

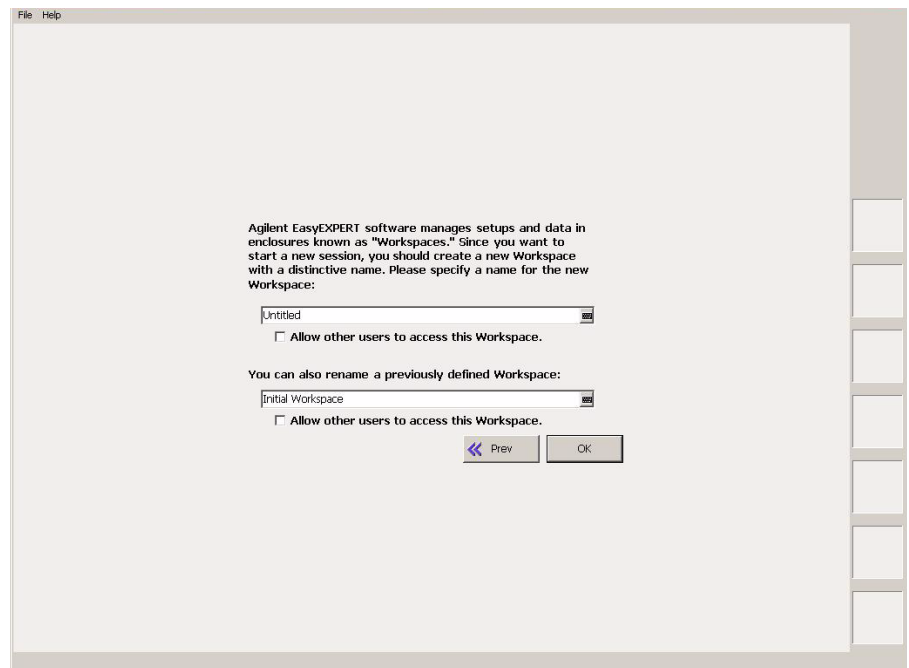
The Next button will work to display the following entry fields used to set the workspace name.

- Entry field for the name of the new workspace
- Entry field for the new name of the existing workspace  
This field is available only for the owner of the existing workspace.

Check the *Allow other users to access this workspace* box to create a *public* workspace that is opened for all users.

The OK button creates and/or renames the workspace, and opens the EasyEXPERT main screen.

The Prev button cancels the workspace creation and returns the screen message to the previous one.



## Main GUI

### Workspace Configurator

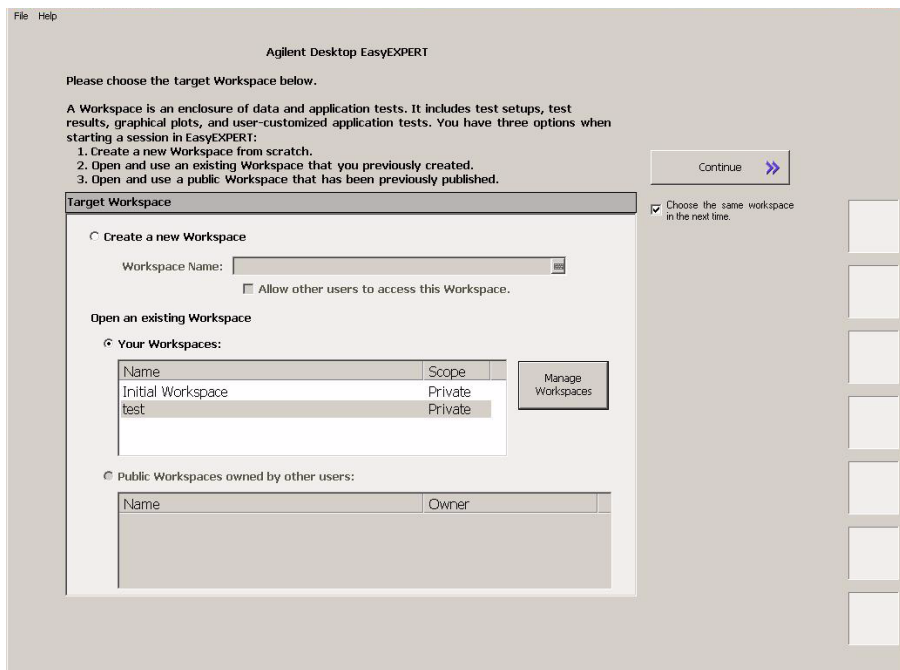
- *No, I want to manage Workspaces.*

This button must be selected if you want to manage workspaces. If you select this button, the label of the OK button will change to Next.

The Next button will work to change the screen as shown in “If there are multiple workspace”.

## If there are multiple workspace

You will see the screen as shown in the following example.



- Continue

This button displays the main screen with the specified workspace.

- *Choose the same Workspace in the next time.*

If this check box is checked, the workspace selection screen will be skipped at the next startup, and EasyXPRT will be launched with the workspace used at the last operation. To perform this setup again, click the File > Close Workspace menu on the EasyXPRT main screen.

- Create a new Workspace

This button must be selected if you want to create a new workspace. Enter the name of the new workspace in the Workspace Name field.

Check the *Allow other users to access this workspace* box to create a *public* workspace that is opened for all users.

- Open an existing Workspace

- Your Workspace

This button must be selected if you want to open your workspace. Select the name to specify the workspace to open.

- Name: Workspace name
- Scope: Private (only for owner) or Public (for all users)

The Manage Workspaces button opens Workspace Manager which provides the buttons used to manage workspaces. Select the workspace name and click a button.

- Delete: Deletes the selected workspace.
- Rename: Renames the selected workspace.
- Duplicate: Makes a copy of the selected workspace.
- Import: Opens a dialog box used to import a workspace.
- Export: Opens a dialog box used to export the selected workspace. The workspace is exported as a file of the extension *xws*.
- Public: Changes the scope of the selected workspace to Public.
- Close: Closes the Workspace Manager.

- Public Workspace owned by other users

This button must be selected if you want to open a public workspace created by another user. Select the name to specify the workspace to open.

- Name: Workspace name
- Owner: Owner of the workspace

---

**NOTE**

**Private workspace**

Private workspace is secured by the user account of its owner. User accounts other than the owner cannot open the Private workspace.

User account of owner: <domain name>\<user name>

Example: B1500A-12345678\Agilent B1500 User

If you do not join the domain but login the local computer, <domain name> will be the computer name.

---

**NOTE**

**If you change the user account**

If you need to change the domain name, the computer name, or the user name, change the scope of all workspace in the EasyEXPERT database to Public before changing the user account.

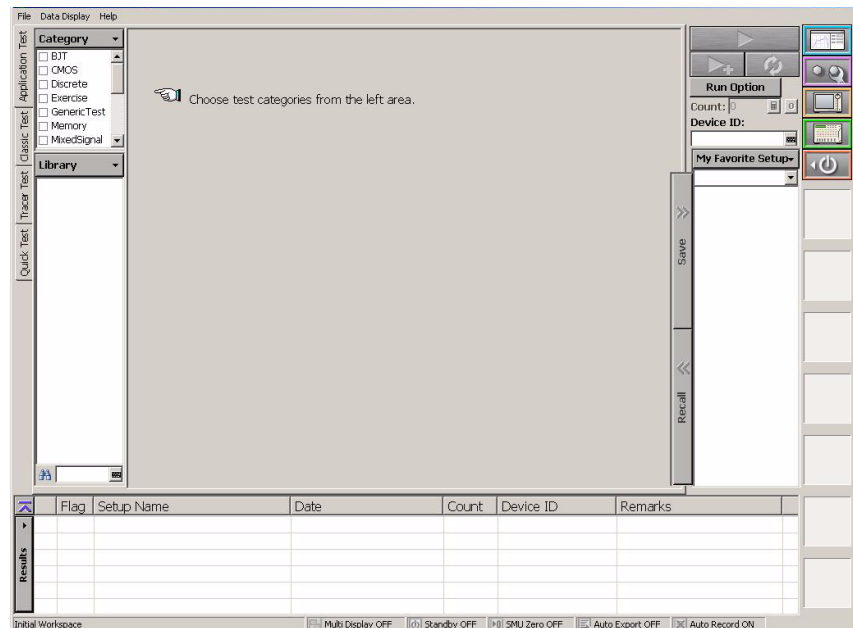
The new account cannot open the Private workspace created by the previous user account.

Also, it is recommended to make a backup of the EasyEXPERT database before changing the user account after changing the scope to Public.

## Main Screen

The EasyEXPERT main screen will be displayed after the workspace is specified or created by the workspace configurator. The main screen provides several tools and functions described in this section.

- “Main Screen GUI”
- “Run Option”
- “Data Display Manager”
- “Data Display Properties”
- “Test Result Editor”
- “Test Results Data Filter”
- “Export in My Format”
- “Test Results Data Folder Export”
- “Test Results Data Auto Export”
- “Test Results Data Properties”



## Main Screen GUI

### File menu

Provides the following four functions.

<b>Import Test Setup...</b>	Opens a dialog box used to specify a test setup to import.
<b>Export Test Setup...</b>	Opens a dialog box used to specify a destination file for exporting the present test setup.
<b>Save Data As...</b>	Only for the tracer test. Opens the Save Data As... dialog box used to specify the file name of the measurement result data to save. Available data formats are the Text, CSV, XML spreadsheet, BMP, EMF, GIF, and PNG.
<b>Close Workspace</b>	Closes the work space and returns to the workspace configurator.
<b>Exit</b>	Exits the EasyEXPERT software.

**Data Display menu** Provides the following three functions.







<b>Show Data Display</b>	Opens “Data Display (page 1-84)” used to display the test result data.
<b>Manage Data Display...</b>	Opens “Data Display Manager (page 1-20)” used to control the appearances of the Data Display windows.
<b>Default Data Display Properties...</b>	Opens “Data Display Properties (page 1-21)” used to set the default setting of the Data Display window.

### Option menu

Provides the following two functions. Only for the tracer test.

<b>Oscilloscope View</b>	Opens/closes the Oscilloscope View window. For details, see “Oscilloscope View” on page 1-42.
<b>Arithmetic Operation</b>	Displays the Arithmetic Operation area, and enables/disables the expressions defined in that area respectively. For details, see “Arithmetic Operation Area” on page 1-47.



- Help menu** Provides the following three functions.
- About This Instrument...** Opens the Configuration window.
  - About This Test...** Opens the test description dialog box.
  - Agilent EasyEXPERT Help** Opens the EasyEXPERT help window.
- Application Test** This tab selects the application test mode of EasyEXPERT. For details of the graphical user interface in this mode, see “Application Test” on page 1-29.
- Classic Test** This tab selects the classic test mode of the EasyEXPERT. For details of the graphical user interface in this mode, see “Classic Test Definition” on page 2-1.
- Tracer Test** This tab selects the tracer test mode of EasyEXPERT. For details of the graphical user interface in this mode, see “Tracer Test” on page 1-31.
- Quick Test** This tab selects the quick test mode of the EasyEXPERT. For details of the graphical user interface in this mode, see “Quick Test” on page 1-48.
- Data Display**  Opens “Data Display (page 1-84)” used to display the test result data. EasyEXPERT always handles the test setup corresponding to the Data Display window now active or most recently activated.
- Calibration**  Opens “Calibration (page 1-58)” used to see the self-calibration results, to perform the self-calibration, or to perform the current offset cancel.
- Configuration**  Opens “Configuration (page 1-64)” used to see the module configuration, to see the self-test results, to perform self-test, to establish the GPIB connection with Agilent B2200A/B2201A/E5250A switching matrix, or to see the event log.
- Switching Matrix**  Not available for the B1505A. Opens “Switching Matrix Operation Panel (page 1-81)” used to control Agilent B2200A/B2201A/E5250A switching matrix.
- Standby**  Opens “Standby Channel Definition (page 1-83)” used to set the standby channel which is the channel to keep the specified output when the standby mode is ON and in the non-measurement state.
- Single**  Triggers a single measurement. To abort measurement, click Stop button that appears instead of the Single button.

**Append**



Triggers the append measurement. The measurement results will be appended to the Data Display window that shows the previous measurement results. The Data Display window can have maximum 10 layers for displaying measurement results. To abort measurement, click Stop button that appears instead of the Single button.

**Repeat**



This button provides a different function depending on the status of the data record functions (Auto Record and Auto Export). See “Run Option (page 1-19)” for the data record status.

If Auto Record and Auto Export is OFF, this button starts the repeat measurement immediately.

If Auto Record or Auto Export is ON, this button opens the “Repeat Measurement Setup (page 1-50)” dialog box for setting the repeat measurement condition.

**Run Option**

Opens the “Run Option (page 1-19)” dialog box for setting the data record functions (Auto Record and Auto Export) and the Multi Display function.

**Save data**



This button will appear to the right side of the Run Option button if a measurement is performed when the data record functions (Auto Record and Auto Export) are OFF. Clicking the button will save the last measurement data as a test result record.

**Count**

Displays the number of single measurements executed for the test setup now opened. Effective for the measurements triggered by both Single and Repeat.



The 0 button sets the Count field to zero.

**Device ID**

This entry field is used to enter a string in the test record. The string can data ID, device ID, test ID, or anything you want.

Performing measurement and changing setup do not change the value in this field. Change the value if you want.

**My Favorite Setup**

The test setups you create or modify MUST be saved to My Favorite group (preset group) to reuse them.

Click the My Favorite Setup button to display the menu that provides the following functions. Select the preset group by using the field below the button. The test setups can be recalled by using the area below the field.

**Cancel** Closes the menu.

**Preset Group** Provides the following seven functions.

**Add New Preset Group** Opens the Add Preset Group dialog box. Creates a new preset group and saves it as the specified name.

<b>Rename This Preset Group</b>	Opens the Rename Preset Group dialog box. You can change the name of the present preset group.
<b>Duplicate This Preset Group</b>	Opens the Duplicate Preset Group dialog box. Creates a copy of the present preset group and saves it as the specified name.
<b>Delete This Preset Group</b>	You can delete the present preset group.
<b>Import Preset Group...</b>	Opens the Preset Group Import dialog box. Imports the specified preset group.
<b>Export This Preset Group...</b>	Opens the Preset Group Export dialog box. Exports the present preset group as the specified name.
<b>Organize Preset Group...</b>	Opens the “Organize Preset Group (page 1-56)” dialog box used to organize the preset group.
<b>Rename</b>	You can change the name of the selected test setup.
<b>Duplicate</b>	Creates a copy of the selected test setup.
<b>Delete</b>	Deletes the selected test setup.
<b>Delete All</b>	Deletes all test setups in the present preset group.

**Save** The button saves the present test setup into the present preset group as a new test setup of the group.

**Recall** The button recalls the selected test setup. This means opening and displaying the measurement definition of the selected test setup.

---

**NOTE** **If a tracer test setup is recalled on the test mode other than the tracer test**

The tracer test setup is converted to and recalled as a Multi Channel I/V Sweep test setup.


In this conversion process, the VAR1 dual polarity sweep setup (Dual Polarity) is ignored. The setup of the positive sweep from start to stop is converted, but the setup of the negative sweep from –start to –stop is not converted.


---

**Flag** Shows the flag, Important (!), Valid (#), or Questionable (?) set to the record. The top button sorts the test records by the flag.

While the “Show Deleted Data (page 1-17)” function is enabled, the list area can show the test records in Delete-group. For the records, \* is set to the column to the left of the Flag column. The top button sorts the list by \*.

<b>Setup Name</b>	Shows the test setup name. The top button sorts the test records by name, in lexicographical order.
<b>Date</b>	Shows the date that the test result is recorded. The top button sorts the test records by date, in chronological order.
<b>Count</b>	Shows the measurement count shown in the Count field. The top button sorts the test records by count, in numerical order.
<b>Device ID</b>	Shows the string set to the record by using the Device ID field. The top button sorts the test records by device ID, in lexicographical order.
<b>Remarks</b>	Shows the remarks set to the record. The top button sorts the test records by the remarks in the lexicographic order.
<b>Thermometer OFF</b>	<p>This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the Thermometer is enabled. And one of the following informations is displayed instead of “Thermometer OFF”. The information is toggled by clicking the indicator. Where, XX indicates the temperature value.</p> <ul style="list-style-type: none"><li>• T1: XX deg C Temperature measured by the thermometer channel 1.</li><li>• T2: XX deg C Temperature measured by the thermometer channel 2.</li><li>• dT: XX deg C Difference between temperature values measured by the channels 1 and 2.</li></ul>
<b>Multi Display ON</b>	<p>This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the Multi Display function is enabled.</p>
<b>Standby ON</b>	<p>This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the standby function is enabled (standby mode is ON).</p>
<b>SMU Zero ON</b>	<p>This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the SMU current offset cancel function is enabled.</p>

**Auto Export ON**  This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the Auto Export (automatic data export) function is enabled.

**Auto Record OFF**  This indicator is located at the bottom of the EasyEXPERT main screen and is lit while the Auto Record (automatic data record) function is disabled.

**workspace name** Field below the Results button shows the present workspace name.

**Results** This button provides the following functions. The list area shows the test records. The up or down arrow button is used to expand or reduce the list area.

- |  |   |
|--|---|
| <b>Cancel</b>                          | Closes the menu.  |
| <b>Select All</b>                      | Selects all records.  |
| <b>Unselect All</b>                    | Unselects all records.  |
| <b>Display Data</b>                    | Opens the “Data Display (page 1-84)” and displays the test result data of the selected record. Up to 15 windows can be opened by this function.   |
| <b>Recall</b>                          | Recalls the test setup of the selected record.  |
| <b>Filter</b>                          | Provides the following four functions.  |
| <b>Show Deleted Data</b>               | Check this function to see the test records of both Save-group and Delete-group. To set the data group, see “Flag and Remarks” on page 1-19. See also “Test Result Editor (page 1-23)”.   |
| <b>Filter...</b>                       | Opens the “Test Results Data Filter (page 1-23)” dialog box. You can filter the test records in the list area.  |
| <b>Expand Application Test Results</b> | Check this function to see all test records created by the test setups defined in an application test.<br><br>If a test record is exported when this function is not checked, the exported file will contain all records created by the test setups defined in an application test. However, if the test record is exported by the Export As Text File function, the exported file will contain only the test record of the application test. |

<b>Show All Append Data</b>	Check this function to see all test records created by the append measurement. In the test record list, <b>S</b> indicates the test records obtained by a single measurement, and <b>A</b> indicates the records obtained by an append measurement.
<b>Transport Data</b>	Provides the following ten functions.
<b>Import...</b>	Opens the Test Result Import dialog box. Imports the specified test record.
<b>Export As Test Result...</b>	Opens the Test Result Export dialog box. Exports the selected test record as the specified name in the EasyEXPERT data format.
<b>Export As Compressed Test Result...</b>	Opens the Compressed Test Result Export dialog box. Exports the selected test record as the specified name in the compressed EasyEXPERT data format.
<b>Export As CSV...</b>	Opens the CSV File Export dialog box. Exports the selected test record as the specified name in the CSV format that can be read by a spread sheet software.
<b>Export As XML Spread Sheet...</b>	Opens the XML Style Sheet Export dialog box. Exports the selected test record as the specified name in the XML spread sheet format that can be read by a spread sheet software.
<b>Export in My Format...</b>	Opens the “Export in My Format (page 1-24)” dialog box. Exports the selected test record as the specified name in the style converted by using the specified XSLT filter.
<b>Export As Text File...</b>	Opens the Text File Export dialog box. Exports the selected test record as the specified name in the style defined on the “Text File Export settings (page 1-111)” dialog box.
<b>Text File Export Setting...</b>	Opens the “Text File Export settings (page 1-111)” dialog box used to define the style of the test record exported by the Export As Text File function.
<b>Folder Export...</b>	Opens the “Test Results Data Folder Export (page 1-24)” dialog box used to export the selected multiple test records to the specified folder.

<b>Auto Export Setting...</b>	Opens the “Test Results Data Auto Export (page 1-26)” dialog box used to set the automatic data export function and the automatic data save function.
<b>Edit</b>	Provides the following two functions.
<b>Flag and Remarks</b>	Opens the “Test Result Editor (page 1-23)” used to put the selected test record in Save-group or Delete-group. Also this dialog box can set a flag and remarks to the selected record. The available data flags are Important (!), Valid (#), and Questionable (?).
<b>Delete</b>	Puts the selected test records in Delete-group. The records can be displayed in the list area while the “Show Deleted Data (page 1-17)” is checked.
<b>Properties...</b>	Opens the “Test Results Data Properties (page 1-28)” dialog box used to set the number of test records listed in the list area and the number of recoverable records in Delete-group.

## Run Option

This dialog box appears by clicking the Run Option button and is used to set the data record functions (Auto Record and Auto Export) and the Multi Display function. This dialog box provides the following GUI.

- Record Test Result Data Automatically

This check box changes the status of the data record functions (Auto Record and Auto Export). The functions provide four status shown in Table 1-1. Removing the check sets Auto Export=Auto Record=OFF. And putting the check again returns the status to the previous one.

In the checked status, the data record functions can be set to one of three status except for Auto Export=Auto Record=OFF. To set the functions, use “Test Results Data Auto Export (page 1-26)”.

In the unchecked status (Auto Export=Auto Record=OFF), the test result data will not be recorded. If a measurement is performed in this status, the Save Data button will appear to the right side of the Run Option button. Clicking the Save Data button will save the last measurement data as a test result record.

When the automatic data record function is enabled, one test execution result is stored in one test record. For the classic test, one test result is stored in one test record. For the application test, all test results taken by executing one application test are stored in one test record. However, the automatic data export function using the Text File type exports the file for each test result.

**Table 1-1 Record Test Result Data Automatically Check Box and Data Record Functions**

Checked status		Unchecked status	
Auto Export	Auto Record	Auto Export	Auto Record
ON	OFF	OFF	OFF
ON	ON		
OFF	ON		

- Allocate Data Display for each test

This check box sets the Multi Display function ON (checked) or OFF. This function controls the operation of the Data Display window.

**OFF** Test result data will be always displayed on the singular Data Display window even if the test setup is changed.

**ON** Test result data of the same test setup name will be displayed on the exclusive Data Display window and the test result data of the different test setup name will be displayed on the new Data Display window.

## Data Display Manager

This dialog box is opened by selecting the Manage Data Display... function of the Data Display menu, and is used to control the appearances of the Data Display windows. This dialog box provides the following three action buttons.

**OK** Applies the setup changes and closes this dialog box.

**Cancel** Cancels the setup changes and closes this dialog box.

**Apply** Just applies the setup changes.



### Data Display Window Order

This area lists the Setup Name of the Data Display windows opened. The list items from top to bottom correspond to the windows from front to back in the screen image. This area supports the following buttons. To select the item, click one in the list.

- Up** Moves the selected item upward.
- Down** Moves the selected item downward.
- Select All** Selects all items.
- Minimize** Minimizes the windows corresponding to the selected items. The list items move to the Minimized Data Displays area.
- Close** Closes the windows corresponding to the selected items.

### Minimized Data Displays

This area lists the Setup Name of the Data Display windows minimized. This area supports the following buttons. To select the item, click one in the list.

- Select All** Selects all items.
- Restore** Restores the windows corresponding to the selected items.
- Close** Closes the windows corresponding to the selected items.

### Data Display Window Configuration

This area provides the following radio buttons and a check box to change the appearances of the Data Display windows. All functions are available for the opened windows.

- Keep Setup** Keeps the present configuration of the Data Display windows.
- Tiling** Resize and collate the data display windows such that they fill up the entire screen.
- Stacking** Resize and collate the data display windows so that they overlap slightly with an offset.
- Overlaying** Resize, overlap, and make transparent the data display windows so that multiple graphs can be viewed and superimposed on one another.

### Apply same scale to all graph

This check box must be checked to apply the same graph scale to all graphs.

## Data Display Properties

This dialog box is opened by selecting the Default Data Display Properties... function of the Data Display menu, and is used to set the default setting of the Data Display window. Click the OK button to apply the setup and close this dialog box.

## Effective Area

This area is used to select the area effective when the Data Display window is opened, and provides the following check box.

- X-Y Graph**            Enables the X-Y Graph area.
- Logo ON**                Displays the Agilent logo at the upper left of the graph area.
- Grid ON**                Displays the grid on the graph.
- Setup Summary ON**        This is valid only for the I/V Sweep and Multi Channel I/V Sweep classic tests. Displays the channel setup summary at the bottom of the graph area. See “Setup Summary” on page 1-101.
- List Display**            Enables the List Display area.
- Parameters**            Enables the Parameters area.

## Graph Color

This area is used to set the color map for the X-Y Graph Plot area. You can change the color of the background, text, graph grid, and line/cursor. The color palette is opened by clicking the pattern of the item.

## Graph Trace Color

This area is used to set visibility, color, and thickness of the data traces Y1 to Y8 plotted on the graph. The color palette is opened by clicking the pattern of the item.

## Log Scale

This area is used to set the maximum number of decades to show the minor ticks for on the log axis. If the number of graph decades is less than or equal to the entered value, the minor ticks are displayed. If it is greater than the entered value, the minor ticks are not displayed.

## Text Size

This area is used to set the font size for the graph. The available settings are Largest, Large, Medium, Small, Smallest, and Fixed. For settings other than Fixed, the appropriate size is automatically selected. For Fixed, a font size entry field is provided, and the specified font size is always applied.

- X/Y Axis Title**        Font size for the axis title
- Scale**                    Font size for the X and Y coordinate values
- Legend**                 Font size for the graph legend

## Test Result Editor

The Test Result Editor appears when measurement is completed or by clicking Results > Edit > Flag & Remarks. This is used to set a flag and remarks to the associated test record and provides the following GUI.

### Save, Delete

Divides test records into groups, Save and Delete.

Test records in Save-group are always listed in the lower area of the EasyEXPERT main screen.

Test records in Delete-group can be listed while Results > Filter > Show Deleted Data is checked.

### Flags

The following buttons are available. You can set one of the following flags.

- Important ! button  
Sets the important flag (!) to the test record.
- Valid # button  
Sets the valid flag (#) to the test record.
- Questionable ? button  
Sets the questionable flag (?) to the test record.

### Remarks

You can enter characters into this field. The characters will be recorded as the Remarks value of the test record.

### OK

Applies the setup on the Test Result Editor, and closes this dialog box.

## Test Results Data Filter

This dialog box appears by clicking Results > Filter > Filter... and is used to specify the test records listed in the lower area of the EasyEXPERT main screen.

### Data records in Save-group

Check this box to list the test records in Save-group.

### Data records in Delete-group

Check this box to list the test records in Delete-group.

### Flag

Select or enter a flag in this field to list the test records with the specified flag.

<b>Setup Name</b>	Select or enter a test name in this field to list the test records of the specified name.
<b>Date</b>	Specify the date to list the test records within the specified period.
<b>Count</b>	Specify the count values to list the test records within the specified range.
<b>Device ID</b>	Select or enter the device ID to list the test records of the specified device ID.
<b>Remarks</b>	Specify the remarks value to list the test records with the specified value.
<b>OK</b>	Performs filtering, and closes this dialog box.
<b>Cancel</b>	Cancels filtering, and closes this dialog box.

## **Export in My Format**

This dialog box appears by clicking Results > Transport Data > Export in My Format..., and is used to export the test record by performing the style conversion using the XSLT filter.

<b>My Format</b>	Specify the full path name of the XSLT filter (XML style sheet) used to convert the style of the test record. The Browse... button opens the Select XML Style Sheet File dialog box used to select the file.
<b>Export As...</b>	Opens the Export As dialog box. Specify the file name and the file type, and export the test record.
<b>Cancel</b>	Cancels data export, and closes this dialog box.

## **Test Results Data Folder Export**

This dialog box appears by clicking Results > Transport Data > Folder Export... after the test result records are selected (highlighted).

This function is used to export the multiple test records. The test records can be exported by specifying the records, the destination folder name, and the naming rule and the file type for the exported files.

<b>Destination for Export</b>	Selects the folder used to save the exported test records. The Browse... button opens the Browse For Folder dialog box used to select the folder.
<b>File Name</b>	Specifies the style of the file name for the exported test records.

File name is automatically generated by using Setup Name, Device ID, Count, and Date values defined in the main screen. If an impermissible character is used in a value, the character is automatically replaced with \_ (underscore).

- Default Format: File name without extension is as follows.

*Flag SetupName [DeviceID(Count) \_Remarks\_ ; Date]*

Example: # Id-Vd [ cmos1(1) \_ok\_ ; 1\_22\_2006 3\_44\_55 PM]

- Custom Format: You can specify the style by the File Name Format dialog box.

The File Name Format dialog box appears by clicking the Settings... button. In the dialog box, enter the desired style into the Format field, for example, the next example sets the same style as the Default Format.

Example: {4} {0} [ {3} ( {2} ) {5} ; {1} ]

{0}: Setup Name

{1}: Date, mm\_dd\_yyyy hh\_mm\_ss AM/PM

{2}: Count

{3}: Device ID

{4}: Flag

{5}: Remarks

## File Type

Selects the file type (data format).

<b>Test Result</b>	EasyEXPERT data format
<b>Compressed Test Result</b>	Compressed EasyEXPERT data format
<b>CSV</b>	CSV format
<b>XML Spread Sheet</b>	XML spread sheet format
<b>My Format</b>	Custom format created by the specified XSLT filter. The following two entry fields are available.

- My Format

Specifies the file name of the XSLT filter (XML style sheet) used to export test records. The Browse... button opens the Select XML Style Sheet File dialog box used to select the file.

- Extension of Output File

Enters the appropriate extension, or selects the extension from xml, txt, csv, or xls.

**Text File** Custom format defined in the “Text File Export settings (page 1-111)” dialog box. The Settings... button opens this dialog box.

**Export** Starts the folder export. This dialog box is closed after the folder export is completed.

**Cancel** Cancels the folder export, and closes this dialog box.

## Test Results Data Auto Export

This dialog box appears by clicking Results > Transport Data > Auto Export Setting... and is used to set the data record functions (Auto Record and Auto Export).

There is the following check box at the top of this dialog box.

- *Enable automatic data export to the specified storage device*

Checking this box sets the Auto Export function (automatic data export) and enables the following GUI.

**Destination for Export** Selects the folder used to save the exported test records. The Browse... button opens the Browse For Folder dialog box used to select the folder.

**File Name** Specifies the style of the file name for the exported test records.

File name is automatically generated by using Setup Name, Device ID, Count, and Date values defined in the main screen. If an impermissible character is used in a value, the character is automatically replaced with \_ (underscore).

- Default Format: File name without extension is as follows.

*Flag SetupName [DeviceID(Count) \_Remarks\_; Date]*

Example: # Id-Vd [cmos1(1) \_ok\_; 1\_22\_2006 3\_44\_55 PM]

- Custom Format: You can specify the style by the File Name Format dialog box.

The File Name Format dialog box appears by clicking the Settings... button. In the dialog box, enter the desired style into the Format field, for example, the next example sets the same style as the Default Format.

Example: {4} {0} [{3}({2}) {5}; {1}]

- {0}: Setup Name
- {1}: Date, mm\_dd\_yyyy hh\_mm\_ss AM/PM
- {2}: Count
- {3}: Device ID
- {4}: Flag
- {5}: Remarks

## File Type

Selects the file type (data format).

- Test Result** EasyEXPERT data format
- Compressed Test Result** Compressed EasyEXPERT data format
- CSV** CSV format
- XML Spread Sheet** XML spread sheet format
- My Format** Custom format created by the specified XSLT filter. The following two entry fields are available.

- My Format
  - Specifies the file name of the XSLT filter (XML style sheet) used to export test records. The Browse... button opens the Select XML Style Sheet File dialog box used to select the file.
- Extension of Output File
  - Enters the appropriate extension, or selects the extension from xml, txt, csv, or xls.

**Text File** Custom format defined in the “Text File Export settings (page 1-111)” dialog box. The Settings... button opens this dialog box.

## Data Storage

Remove a check from the following check box to disable the Auto Record function (automatic data record).

- *Enable automatic data record to the internal storage device*

## OK

Applies the setup, and closes this dialog box.

## Cancel

Cancels the setup, and closes this dialog box.

---

## NOTE

**Automatic Data Record Output**

When the automatic data record function is enabled, one test execution result is stored in one test record. For the classic test, one test result is stored in one test record. For the application test, all test results taken by executing one application test are stored in one test record. However, the automatic data export function using the Text File type exports the file for each test result.

---

## Test Results Data Properties

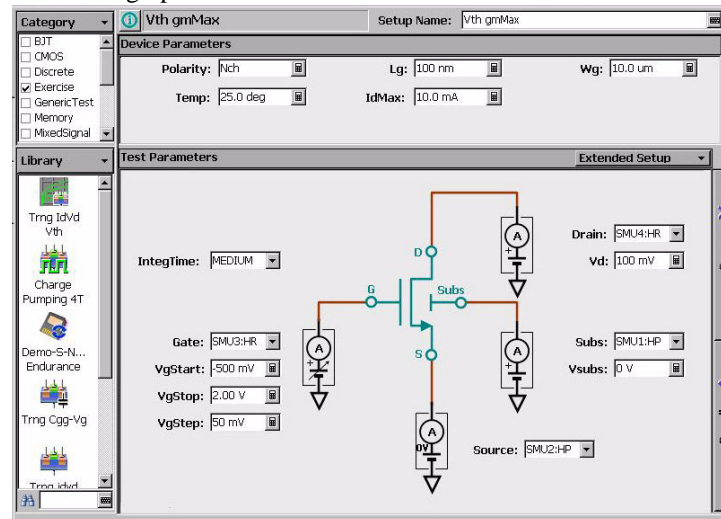
This dialog box appears by clicking Results > Properties... and is used to set the maximum number of test records listed in the main screen and saved in the Delete-group.

<b>Test Result List View</b>	Sets the maximum number of test records listed in the main screen.
<b>Non-preserved data</b>	Sets the maximum number of test records saved in the Delete-group.
<b>Close</b>	Closes this dialog box.



## Application Test

In the application test mode, the EasyEXPERT main screen has the following additional graphical user interface.



### Category

Lists the categories of the application test definitions. You can select the categories by using the check box. The test definitions listed in the Library area depend on the selection of the categories.

The Category button opens the Categories dialog box used to select the categories.

### i, test name

This field just displays the test definition name of the present application test. Clicking the left **i** button displays the help message for the test definition.

### Library

Lists the application test definitions that belong to the categories selected by the Category area.

Clicking an icon displays its brief introduction message and the Select button. Clicking the Select button loads the application test definition. Now, the B1500 is ready to start measurement.

The Library button provides the following six functions.

**Close** Closes this menu.

**Define New Test...** Opens the “Test Definition Window (page 3-3)” used to create a new application test.

**Open Definition of This Test...**

Opens the “Test Definition Window (page 3-3)” to display the definition of the present (selected) application test or to modify the definition.

**Delete Definition of This Test** Deletes the present (selected) application test. The test will be deleted from the application library.

**Import Test Definition...** Opens a dialog box used to specify the test definition to import.

**Export Test Definition...** Opens a dialog box used to export the present application test definition. The test definition will be saved to the specified file.

**Search**



Incremental search. Used to search the application test definitions listed in the Library area.

**Setup Name**

Used to set the name of test setup. The name must be the identification for the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is stored in the test result data and is used to differentiate the records in the test record list area.

**Device Parameters**

This area displays the device parameter entry fields if the device parameters are defined as the variables in the application test definition.

Enter the appropriate value to the entry fields before starting measurement.

**Test Parameters**

This area displays the device connection diagram if it is specified in the application test definition.

This area also displays the test parameter entry fields if the test parameters are defined as the variables in the application test definition. Enter the appropriate value to the entry fields before starting measurement. If the entry field needs to enter a vector data, see “Define vector data” on page 2-18.

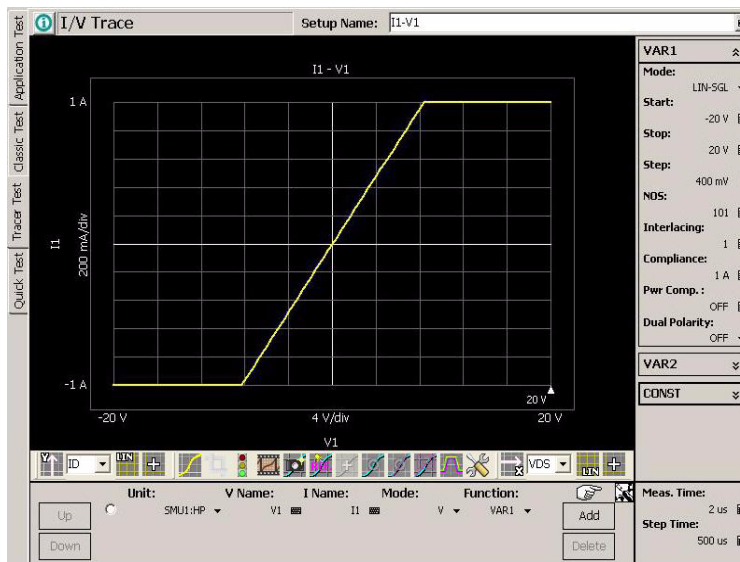
**Extended Setup**

This button opens the Extended Setup dialog box that displays the extended parameter entry fields. See “Ext” on page 3-6 for the parameters shown in this dialog box.

Enter the appropriate value to the entry fields and click Close button.

## Tracer Test

In the tracer test mode, you can start I/V sweep measurement easily and quickly by setting up the I/V Trace screen only. The EasyEXPERT main screen has the following additional graphical user interface.



### Setup Name

Used to set the name of test setup. The name must be the identification for the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is recorded in the test result data and is used to differentiate the data in the test record list area.

### Graph area

This area displays measurement result graph and provides “Graph Setup Tool” and “Option Tool” for setting and changing the graph display.

Mouse drag operation zooms in the graph. To set the graph maximum or minimum value directly, use the entry field displayed by clicking the value on the graph.

### Channel area

This area defines the channel setups and the variables for the channel output data and the measurement data.

## Main GUI

### Tracer Test

<b>Add</b>	Adds a row of the channel setup by selecting a SMU listed in the menu displayed by clicking this button.
<b>Delete</b>	Deletes the selected channel setup.
<b>Up / Down</b>	Moves the selected channel setup upward or downward.
<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	Output mode. V (voltage), I (current), VPULSE (voltage pulse), or IPULSE (current pulse).
<b>Function</b>	Output function, VAR1 (primary sweep), VAR2 (secondary sweep), or CONST (constant)

To select a channel setup, use the left radio button of the channel setup. If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. The output stops in the opposite order.

### Sample setups



Clicking this button displays the sample setup menu. Clicking a setup name applies its setup to the tracer test environment.

The B1500A provides the following sample setups.

- MOSFET: ID-VDS, ID-VGS, ID(off)-VDS, BVDSS
- Diode: IF-VF, IR-VR
- BJT: IC-VCE, IC-VCEO, BVCEO

The B1505A provides the following sample setups.

- MOSFET: ID-VDS, ID-VGS, ID(off)-VDS, BVDSS
- Diode: IF-VF, IR-VR
- IGBT: IC-VCE, IC-VGE, IC(off)-VCE, BVCEs
- BJT: IC-VCE, IC-VCEO, BVCEO, IC-VCBO, BVCBO
- GaN FET (N1267A): ID-VDS, ID(off)-VDS, ID-VDS Current Collapse, FET Current Collapse Oscilloscope View
- GaN Diode (N1267A): IF-VF, IR-VR, IF-VF Current Collapse, Diode Current Collapse Oscilloscope View

## VAR1

This area defines the primary sweep source output. All of the following parameters are displayed for the full mode. Only Mode, Stop, NOS, and Compliance are displayed for the short mode. Click the VAR1 title bar to switch the mode.

<b>Mode</b>	Sweep mode. The single mode sweeps from start to stop, and the double mode sweeps from start to stop to start.								
	<table> <tr> <td><b>LIN-SGL</b></td> <td>Linear sweep, single mode</td> </tr> <tr> <td><b>LOG-SGL</b></td> <td>Logarithmic sweep, single mode</td> </tr> <tr> <td><b>LIN-DBL</b></td> <td>Linear sweep, double mode</td> </tr> <tr> <td><b>LOG-DBL</b></td> <td>Logarithmic sweep, double mode</td> </tr> </table>	<b>LIN-SGL</b>	Linear sweep, single mode	<b>LOG-SGL</b>	Logarithmic sweep, single mode	<b>LIN-DBL</b>	Linear sweep, double mode	<b>LOG-DBL</b>	Logarithmic sweep, double mode
<b>LIN-SGL</b>	Linear sweep, single mode								
<b>LOG-SGL</b>	Logarithmic sweep, single mode								
<b>LIN-DBL</b>	Linear sweep, double mode								
<b>LOG-DBL</b>	Logarithmic sweep, double mode								
<b>Start</b>	Sweep start value								
<b>Stop</b>	Sweep stop value								
<b>Step</b>	Incremental step value of the sweep output. The value is automatically calculated by using the start, stop, and NOS values.								
<b>NOS</b>	Number of sweep steps N. 1 to 1001.								
<b>Interlacing</b>	Number of the interlace sweep M. 1 to 100.  Sweep is repeated M times so that the final sweep step is equal to Step/M. Total number of steps will be $N+(N-1)\times(M-1)$ .								
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source. Measurement is performed within this value.								
<b>Pwr Comp.</b>	Power compliance value (in W), or OFF								
<b>V Compliance</b>	Only for UHCU, UHVU, or HVMCU. Limit value for the voltage sweep output. The output voltage of the measurement resource is automatically measured, and the measured voltage is compared with this limit value. And then, if it exceeds the limit, the measurement status is set to the compliance status, and the sweep measurement stops. Voltage range and setting resolution depend on the measurement resource.								
<b>Dual Polarity</b>	Dual polarity sweep ON or OFF. If this parameter is ON, the VAR1 channel performs positive sweep from start to stop and negative sweep from -start to -stop in this order. For the pulsed negative sweep, the pulse base value is $-1\times(\text{Pulse Base})$ .								

**Hold Time** Hold time (in seconds) of the primary sweep source output just before starting a sweep. 0 to 655.35. Setting resolution: 10 ms. In the hold time, the primary sweep source forces the sweep start level for staircase sweep output, or the pulse base level for pulsed sweep output. At the first step of the primary sweep, the longer one is applied from the VAR1 and VAR2 Hold Time settings.

The following parameters are also displayed for the full mode if the IPULSE or VPULSE is set to the VAR1 channel.

**Pulse Base** Pulse base value  
**Pulse Delay** Pulse delay time. Set 0 for HR/MP/HPSMU.  
**Pulse Width** Pulse width. Must be the same value for all of HR/MP/HPSMU.

## VAR2

This area defines the secondary sweep source output. The VAR2 channel performs the linear single sweep operation. All of the following parameters are displayed for the full mode. Only Start, Stop, Step, and NOS are effective for the short mode. Click the VAR2 title bar to switch the mode.

**Start** Sweep start value  
**Stop** Sweep stop value  
**Step** Incremental step value of the sweep output. The value is automatically calculated by using the start, stop, and NOS values.  
**NOS** Number of sweep steps. 1 to 101.  
**Compliance** Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source. Measurement is performed within this value.  
**Pwr Comp.** Power compliance value (in W), or OFF  
**V Compliance** Only for UHCU, UHVU, or HVMCU. Limit value for the voltage sweep output. The output voltage of the measurement resource is automatically measured, and the measured voltage is compared with this limit value. And then, if it exceeds the limit, the measurement status is set to the compliance status, and the sweep measurement stops. Voltage range and setting resolution depend on the measurement resource.

**Hold Time** Hold time (in seconds) of the primary sweep source output just before starting a sweep. 0 to 655.35. Setting resolution: 10 ms. In the hold time, the primary sweep source forces the sweep start level for staircase sweep output, or the pulse base level for pulsed sweep output. At the first step of the primary sweep, the longer one is applied from the VAR1 and VAR2 Hold Time settings.

The following parameters are also displayed for the full mode if the IPULSE or VPULSE is set to the VAR2 channel.

**Pulse Base** Pulse base value  
**Pulse Delay** Pulse delay time. Set 0 for HR/MP/HPSMU.  
**Pulse Width** Pulse width. Must be the same value for all of HR/MP/HPSMU.

## CONST

This area defines the constant output source.

**Unit** Measurement resource for the constant output. If multiple CONST channels are defined, this field is used to specify the channel for setup.

**Source** Output value.

**Compliance** Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source. Measurement is performed within this value.

**V Compliance** Only for UHCU, UHVU, or HVMCU. Limit value for the voltage sweep output. The output voltage of the measurement resource is automatically measured, and the measured voltage is compared with this limit value. And then, if it exceeds the limit, the measurement status is set to the compliance status, and the sweep measurement stops. Voltage range and setting resolution depend on the measurement resource.

The following parameters are also displayed for the full mode if the IPULSE or VPULSE is set to the CONST channel. Click the CONST title bar to switch the mode. The Pulse Peak entry field appears instead of the Source entry field.

**Pulse Peak** Pulse peak value.  
**Pulse Base** Pulse base value  
**Pulse Delay** Pulse delay time. Set 0 for HR/MP/HPSMU.  
**Pulse Width** Pulse width. Must be the same value for all of HR/MP/HPSMU.

- Meas. Time** This field is used to set the measurement time for a sweep step.
- Meas. Delay Time** This field is used to set the measurement delay time (from the beginning of pulse period to the beginning of measurement) for a pulse step. This entry field appears if a pulse output channel exists.
- 2  $\mu$ s to Pulse period – Measurement time – 2 ms, 2  $\mu$ s resolution
- Measurement delay time = 0 (AUTO) sets the optimum value automatically so that the measurement is completed when the transition from peak to base is started by the pulse output channel which starts the transition at first.
- Step Time** This field is used to set the output time for a sweep step. 500  $\mu$ s to 5 s, 100  $\mu$ s step.
- Pulse Period** This field is used to set the pulse period. This entry field appears instead of the Step Time entry field if a pulse output channel exists.

---

**NOTE**

**About measurement time and measurement error**

In the tracer test mode, measurement is always performed by using the following settings.

- A/D converter: HS ADC (high speed A/D converter)
- Measurement ranging mode: compliance range, fixed
- Effective current measurement range: all for staircase sweep output, 10 nA and greater for bias output or pulse output
- Measurement time: absolute value entered into the Meas. Time field

You may find difference between the results measured in the tracer test mode and in the other test mode. This is caused by the differences of setting values for the above items. The above settings will reduce measurement time and increase measurement error compared with when using HR ADC (high speed A/D converter) and auto ranging in the other test mode.

---

**NOTE**

**Drag the region of VAR1, VAR2, and CONST**

Mouse drag operation will scroll this region when it cannot display all setup parameters.

---



## Graph Setup Tool

The graph setup tool provides the following GUI for setting the graph display.

### X-axis direction



This button is used to set the X-axis direction, left to right (→) or right to left (←).

### Y-axis direction



This button is used to set the Y-axis direction, down to up (↑) or up to down (↓).

### Variable



This field is used to set the X-axis/Y-axis variable name to plot.

Variable must be V Name or I Name used for sweep output or measurement.

### Scale



This button is used to set the X-axis/Y-axis scale, linear (LIN) or logarithm (LOG).

### Polarity



This button is used to set the X-axis/Y-axis polarity, plus (+) only, minus (-) only, or both plus and minus (±). Both plus and minus (±) is available only for the linear scale.

## Option Tool

The following tool is available for changing the graph display.

### Line/Dot



This button is used to set the trace style, line or dot.

### Cancel zoom



This button is available for the graph zoomed in, and is used to return the graph scale to the original setting. The graph can be zoomed in by the point and drag operation.

### Set stop condition area




This button is used to set the abnormal region on the graph, and shows the present status of the abnormal region setup by using the button's signal color. While the abnormal region is effective, the measurement is automatically stopped if the measurement data is in the region.


**Green** Abnormal region has not been defined. Clicking this button changes the button's signal color to Yellow.


**Yellow** Status for defining the abnormal region. Clicking this button changes the button's signal color to Green.


The abnormal region can be defined by the point and drag operation. It will be the square specified by the red grids. After defining the region, this button's signal color is changed to Red.


**Red** Abnormal region is effective. Clicking this button changes the button's signal color to Green.


**Replay traces**  This button opens/closes the dialog box used for managing the record and replay of traces. For the dialog box, see "Replay traces" on page 1-39.


**Capture a reference trace**  This button is used to capture the trace as the reference trace. Clicking this button captures one reference trace. Up to 10 traces can be captured. To manage reference traces, click the reference trace button.


**Select reference traces**  This button opens/closes the dialog box used for managing reference traces memorized by clicking the Capture button. For the dialog box, see "Select reference traces" on page 1-40.

**Select tracking traces**  This button is available for the measurement result which contains at least two traces by the append measurement. Tracking trace means each trace of this measurement result. This button opens/closes the dialog box used for managing tracking traces. For the dialog box, see "Select tracking traces" on page 1-41.

**Marker**  This button enables/disables the marker which can be moved on the active trace by using rotary knob or mouse. The marker position X and Y coordinate values are displayed in the upper left of the graph.

**Tangent line**  This button enables the marker and enables/disables the tangent line for the marker point. The line X-axis intercept, Y-axis intercept, and gradient values are displayed in the upper right of the graph. The tangent line disables the regression line.

**Regression line**  This button enables the marker and enables/disables the line which is given by the regression calculation. The line X-axis intercept, Y-axis intercept, and gradient values are displayed in the upper right of the graph. The regression line disables the tangent line. The measurement data for the regression calculation can be specified by the point and drag operation.

**Oscilloscope View**  This button opens/closes the Oscilloscope View window used for monitoring the waveforms of output signals. For the window, see "Oscilloscope View" on page 1-42.

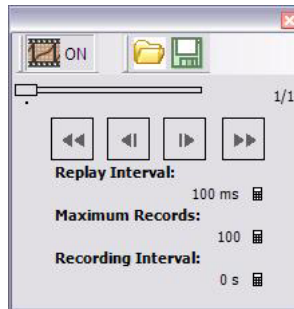
## Color and thickness



This button opens/closes the dialog box used for changing the graph color and trace thickness. For the dialog box, see “Color and thickness” on page 1-41.

## Replay traces

This dialog box is opened/closed by clicking the Replay traces button and is used for managing the record and replay of traces.



## ON / OFF



This button enables/disables the automatic trace record function. While this button labels ON, this function is enabled and continues recording the measurement trace.

## Open



This button opens the Open Trace Record dialog box which is used to open a recorded-trace saved by using the Save Trace Record dialog box.

## Save



This button opens the Save Trace Record dialog box which is used to save the recorded-trace selected on this dialog box. The trace data is saved in the EasyEXPERT Compressed Trace Record format (.gtr).

## Slide bar



The slide bar is used to specify the recorded-trace to display. The trace is specified by  $N/M$  put on the space right next to the slide bar.  $N$  is the record number assigned to the recorded-trace now on graph and  $M$  is the total number of the recorded-traces.

## << (fast back)



Clicking this button changes the displayed recorded-trace to the trace  $N-1$ . Long-click makes this operation fast.

## < (step back)




Clicking this button changes the displayed recorded-trace to the trace  $N-1$ .

## > (step forward)



Clicking this button changes the displayed recorded-trace to the trace  $N+1$ .

**>> (fast forward)** Clicking this button changes the displayed recorded-trace to the trace  $N+1$ .  
 Long-click makes this operation fast.

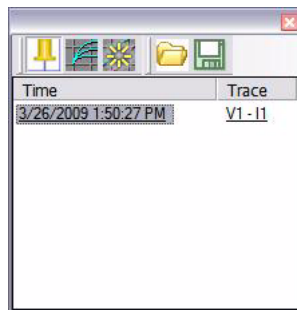
**Replay Interval** This field is used to set the time interval of the fast back and fast forward operation.

**Maximum Records** This field is used to set the maximum record count. After the record number reaches this value, the record numbers are shifted to the newer side and the oldest record is deleted. This number is always assigned to the last record.

**Recording Interval** This field is used to set the time interval of the automatic trace record operation.

## Select reference traces

This dialog box is opened/closed by clicking the Select reference traces button and is used for managing the reference traces memorized by clicking the Capture button.



**Show/Hide selected trace** This button enables/disables the graph display of the trace selected/highlighted on this dialog box.



**Inversion** This button enables/disables the inversion of the trace selected/highlighted on this dialog box.



**Erase** This button deletes the trace selected/highlighted on this dialog box.



**Open** This button opens the Open Reference Trace dialog box which is used to open a reference trace saved by using the Save Reference Trace dialog box.



**Save** This button opens the Save Reference Trace dialog box which is used to save the reference trace selected/highlighted on this dialog box. The trace data is saved in the XML spreadsheet format (.xml).



**Time** This field shows the time stamp of the reference trace.

**Trace** This field shows the title of the reference trace. The title is same as the title appears top of the graph frame.

## Select tracking traces

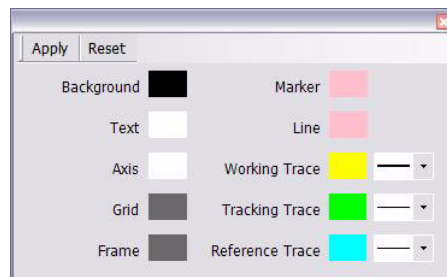
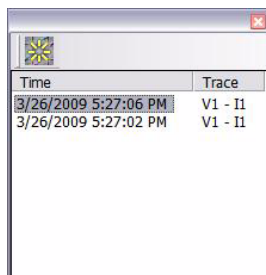
This dialog box is opened/closed by clicking the Select tracking traces button and is used for managing the tracking traces.

**Erase** This button deletes the trace selected/highlighted on this dialog box.



**Time** This field shows the time stamp of the tracking trace.

**Trace** This field shows the title of the tracking trace. The title is same as the title appears top of the graph frame.



## Color and thickness

This dialog box is opened/closed by clicking the Color and thickness button and is used for changing color of the graph color and the trace thickness.

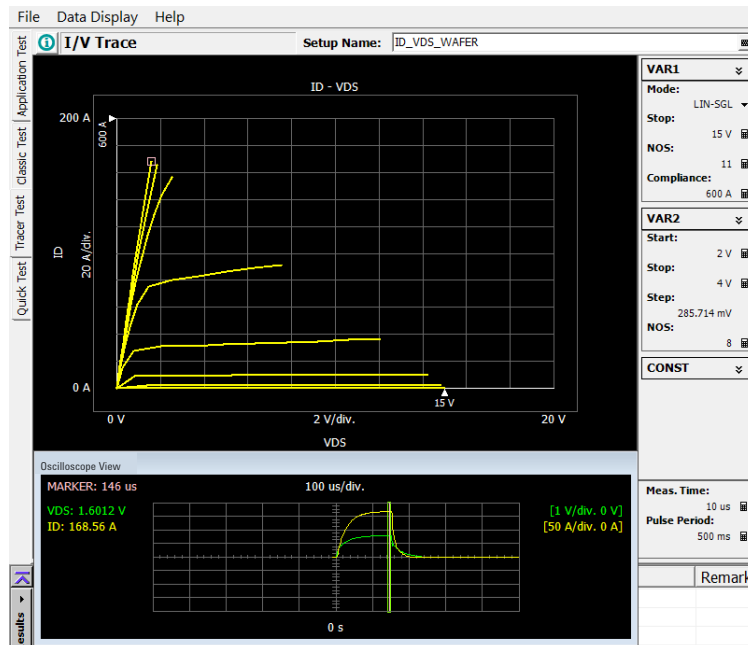
The graph elements supported by this dialog box are background, text, X and Y axes, graph grid, frame around graph, marker, line, and traces.

Clicking the color pattern opens the color palette for selecting color. The color palette provides the function to add the custom colors.

**Apply** This button applies the setup change made on this dialog box to the graph.

**Reset** This button applies the initial setting to this dialog box and the graph.

## Oscilloscope View



This window is opened/closed by clicking the Oscilloscope View button on the Tracer Test screen, or selecting the Option > Oscilloscope View menu.

This function monitors a point of the pulsed sweep measurement or the pulsed spot measurement and displays its waveform. The monitoring point is specified by a rectangular marker on the trace. This function is used for monitoring the time relationship between multiple channel outputs/measurements, the change of the output level for the gate drive channel, and so on.

This function is valid for using one or more VPULSE or IPULSE channels. And this function is not available for the following measurement resources.

- HRSMU, HPSMU, MPSMU
- MFCMU
- WGF MU
- SPGU
- GN DU

Data names set in the V Name and I Name fields of the channel area are used to specify the target monitored signals.

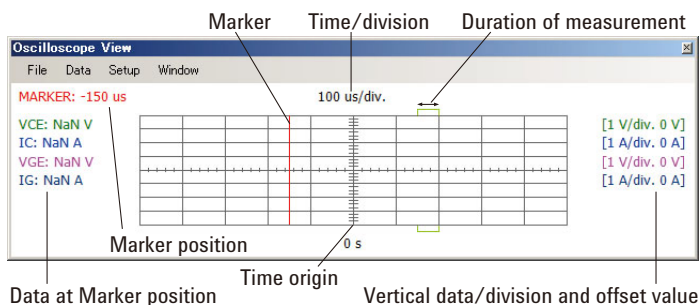
During executing the tracer test, the waveforms within the specified duration are displayed in this window graphically. The display resolution is 2  $\mu$ s.

## Oscilloscope View Window GUI

The Oscilloscope View window provides the following GUI. See Figure 1-1.

Figure 1-1

### Oscilloscope View Window



### File menu

Provides the following function.

- Save Setting** Saves the setup data for waveform monitor. The setup data is saved as a part of the tracer test setup.
- Save Data As...** Opens the Save Waveform Data dialog box used to specify the file name of the displayed waveform data to save.  
Available data formats are the Text (tab separated), CSV, XML spreadsheet, BMP (bitmap image), EMF (enhanced metafile), GIF image, and PNG image.
- Close** Closes the Oscilloscope View window.

### Data menu

Lists all the names of voltage/current output data set in the present tracer test with serial numbers.

Checking a data name enables to monitor the signal corresponding to the data name, and display the waveform in the Oscilloscope View window. If not checking, the waveform for the data name is not displayed in the Oscilloscope View window.

### Setup menu

Provides the following functions.

## Main GUI

### Tracer Test

<b>Time &gt; Scale</b>	Opens the Time Scale dialog box. Enter the scale value of the X axis (time) per division in the dialog box.
<b>Time &gt; Origin</b>	Opens the Time Origin dialog box. Enter the origin value of the X axis (time) in the dialog box.
<b>Sampling</b>	Opens the Sampling Setup dialog box. See “Sampling Setup” on page 1-45.
<b><i>N. data name</i></b>	The data names that are checked on the Data menu are listed. Opens the Data View dialog box. See “Dialog box for changing data display” on page 1-45. Where, <i>N</i> is a serial number.

### Window menu

Provides the following functions.

<b>Large</b>	Changes the window size to the prescribed large window.
<b>Small</b>	Changes the window size to the prescribed small window.

### Monitor area

This area displays a waveform graph in the center. All monitored waveforms are displayed on this graph. A vertical marker is available to read out the values on all waveforms at a certain time.

The waveforms and the relational information are organized by color. To change the color, select the Setup > *N. data name* menu to open the Data View dialog box. See “Dialog box for changing data display” on page 1-45, for this dialog box.

The display items in the monitor area are described below.

#### Marker position

Marker position (X-axis (time) data). Displayed on the upper left of the graph. To move the marker position, use rotary knob or mouse.

#### Data at Marker position

Y-axis (voltage or current) values at the marker position. Displayed with the data names on the left of the graph respectively.



### **Time/division**

Time scale value per division. Displayed on the top of the graph. To change it, click on the displayed value and open the Time Scale dialog box. Enter the scale value of the X axis (time) per division in the dialog box.

### **Time Origin**

Time origin value. Displayed on the bottom of the graph. To change it, click on the displayed value and open the Time Origin dialog box. Enter the origin value of the X axis (time) in the dialog box.

### **Duration of measurement**

Duration of the measurement for the presently-monitored sweep step. Indicated by the green horizontal lines displayed on the top and bottom of the graph. Hence, you can observe the time relationship between the measurement time and the waveform.

### **Vertical data/division and offset value**

Y-axis scale value per division and offset value. Displayed on the right of the graph respectively. To change them, click on the displayed value and open the Data View dialog box. See “Dialog box for changing data display” on page 1-45, for this dialog box.

## **Dialog box for changing data display**

This dialog box is opened by selecting the Setup > *N. data name* menu, or by clicking on the scale or offset value shown on the right area of the graph display. This is used for changing the Y-axis scale, offset, the color of the line and characters, and the line thickness.

Clicking the color pattern opens the color palette for selecting color. The color palette provides the function to add the custom colors.

## **Sampling Setup**

This dialog box is opened by selecting the Setup > Sampling menu. This is used for setting up the sampling conditions of the waveform monitoring.

The target observing step is decided by the following combination:

- The ordinal number of the sweep step on the primary sweep source output (VAR1 Step)
- The ordinal number of the sweep step on the secondary sweep source output (VAR2 Step)

The waveform sampling is started from the beginning of the pulse period within the target observing step. The observing period on the waveforms is eventually decided by setting the sampling delay (Delay) and sampling duration (Duration). Figure 1-2 shows a sampling setup example.

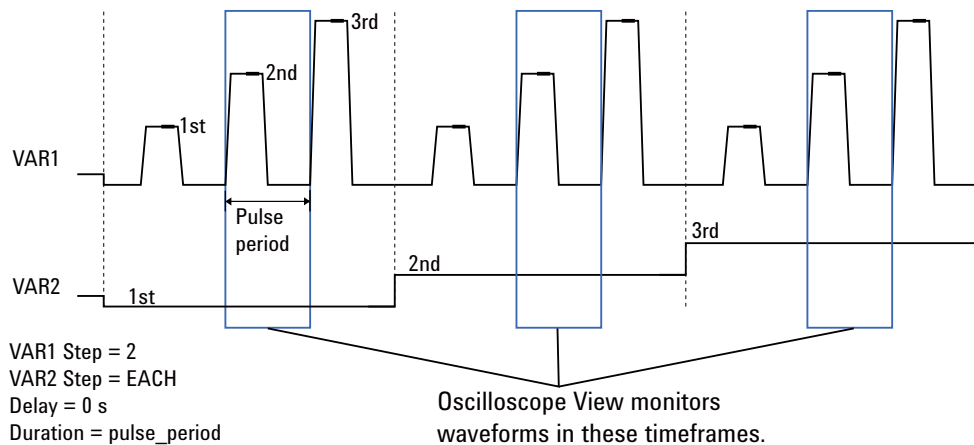
---

**NOTE**

---

If the observing period does not fully cover the measurement time, the measurement data at the sweep step will be "NaN".

**Figure 1-2 Sampling Setup Example**



**VAR1 Step:** The ordinal number ( $N$ ) of the sweep step on the primary sweep source output.  $N$ -th step is specified. For the dual polarity sweep, the ordinal number of the positive sweep step is valid.

For the interlacing sweep, every  $N$ -th step on repeated sweeps is specified.

To specify the last step, enter LAST. Changing the number of sweep step has no impact on the LAST setting.

For the interlacing sweep, every last step on repeated sweeps is specified.

**VAR2 Step:** The ordinal number ( $M$ ) of the sweep step on the secondary sweep source output.  $M$ -th step is specified.

To specify each step, enter EACH.

**Delay:** Sampling delay time. This is the time from the beginning of the pulse period to the beginning of the waveform sampling,  $-30 \mu\text{s}$  to pulse period.

**Duration:** Duration to sample the waveforms,  $22 \mu\text{s}$  to  $24 \text{ms}$ .

## Arithmetic Operation Area

This area appears at the top of the graph by clicking the Option > Arithmetic Operation > Operation  $N$  ( $N$ : 1, 2, or 3) menu, and is used to define arithmetic expressions using the measurement result data. This function is useful to graphically display parameters given by the expressions.

Maximum of three expressions can be defined as Operation 1, Operation 2, and Operation 3. An expression can be defined by two operands and an operator + (addition), - (subtraction), \* (multiplication), or / (division). The following elements can be the operand.

- Measurement data (e.g. `Imeas`)
- Source data (e.g. `Vset`)
- Real literal constant (e.g. `2`, `0.123`, or `100E-3`)
- Operation 1 result (e.g. `Vdrop`), for the expression of Operation 2 and/or Operation 3
- Operation 2 result (e.g. `Vout`), for the expression of Operation 3

### Example

The following example calculates the voltage drop (`Vdrop`) caused by a  $100 \text{k}\Omega$  series resistor and measurement current (`Imeas`), then calculates the voltage (`Vout`) applied to a device under test by subtracting the voltage drop from the setting value (`Vset`).

Operation 1:

$$[ \text{Vdrop} ] = [ 100\text{e}3 ] * [ \text{I<sub>meas</sub>} ]$$

Operation 2:

$$[ \text{Vout} ] = [ \text{Vset} ] - [ \text{Vdrop} ]$$

---

#### NOTE

“NaN” is returned for the invalid calculation results.

---

#### NOTE

To disable the operation, click the Option > Arithmetic Operation > Operation  $N$  menu and uncheck the Operation  $N$  menu.

## Quick Test

In the quick test mode, you can execute the test setups saved in a preset group (*My Favorite Setup*) sequentially. The EasyEXPERT main screen has the following additional graphical user interface.

**My Favorite Setup** The following menu is displayed by clicking this button.

**Cancel** Closes this menu.

**Duplicate This Preset Group** Opens the Duplicate Preset Group dialog box. Creates a copy of the present preset group and saves it as the specified name.

**Import Preset Group...** Opens the Preset Group Import dialog box. Imports the specified preset group.

**Export Preset Group...** Opens the Preset Group Export dialog box. Exports the present preset group as the specified name.

**Organize Preset Group** Opens the Organize Preset Group dialog box used to organize the preset group. See “Organize Preset Group” on page 1-56.

---

### NOTE

#### If the preset group contains a tracer test setup

The tracer test setup is converted to and recalled as a Multi Channel I/V Sweep test setup.

In this conversion process, the VAR1 dual polarity sweep setup (Dual Polarity) is ignored. The setup of the positive sweep from start to stop is converted, but the setup of the negative sweep from -start to -stop is not converted.





---

### NOTE

#### The conversion rule for the hold time settings

The VAR1 hold time setup (Hold Time) is converted and used, and the VAR2 hold time setup (Hold Time) is ignored.

---

- Preset Group**           Selects the preset group to open.
- Setup Name**           Lists all test setups saved in the preset group.
- To select the test setup for the quick test setup operation, click the line. For multiple selection, use ctrl-click, shift-click, or Select All.
- When you are doing the quick test setup, the softkeys are available to start the associated test. The softkey label will show the Setup Name.
- Quick Test List**       The following buttons are available for the quick test setup operation.
- Select All**           Selects all test setup in the test setup list.
- Execution on/off**   The  button is effective for the selected test setup and is used to enable/disable execution of the tests. Once the quick test is started, only the tests marked with  will be executed in order from top to bottom of the test setup list.
- Data save on/off**   The  button is effective for the selected test setup and is used to enable/disable data save of the tests. The data save will be performed only for the tests marked with .
- Up**                    Moves the selected test setup upward
- Down**                Moves the selected test setup downward.
- Recall**               Recalls the selected test setup. The screen is changed to Application Test or Classic Test.
- Rename**              You can change the name of the selected test setup.
- Duplicate**           Creates a copy of the selected test setup and places it at the bottom of the test setup list.
- Delete**              You can delete the selected test setup.

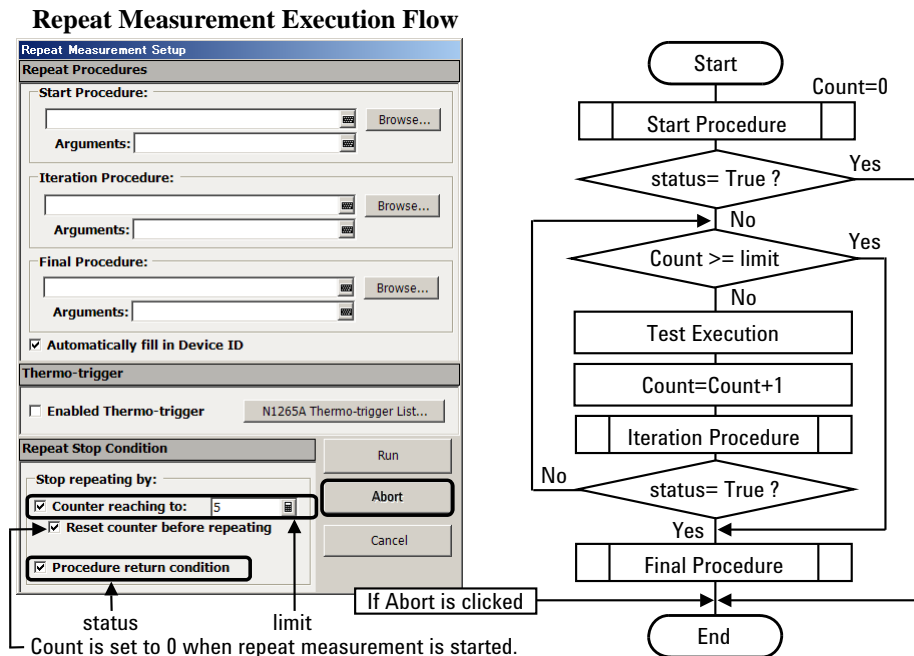
## Repeat Measurement Setup

This dialog box is opened by clicking the Repeat button, and is used to set the repeat measurement condition. This dialog box supports the following action buttons.

- Run** Starts measurement.
- Abort** Aborts the measurement.
- Cancel** Cancels performing the repeat measurement and closes this dialog box.

The repeat measurement is performed as shown in Figure 1-3. The repeat measurement operation is controlled by the value of *Count* and *status*. Where, *Count* is the number displayed in the Count field of the main screen, and *status* must be a parameter of the response created by Start Procedure and Iteration Procedure. For the response, see Figure 1-4.

Figure 1-3



### Repeat Procedures

This area is used to specify the procedure to be called while the repeat measurement is performed. The procedure must be the execution file (.exe file). If no procedure is needed, empty the field. The Browse button opens a file browser.

**Start Procedure** Specifies the procedure executed before starting measurement.

**Iteration**

**Procedure** Specifies the procedure executed every single measurement.

**Final Procedure** Specifies the procedure executed after the repeat measurement.

If the procedure requires arguments, enter the values to the Arguments field.

If the procedure specified by Start Procedure or Iteration Procedure creates the response as shown in Figure 1-4, EasyEXPERT performs the operation following the response. See Figure 1-3. The parameter *status* value must be the string True or False, and the parameter *device\_id* value should be any string. EasyEXPERT ignores the response of the different format.

**Automatically fill in Device ID**

This check box is available when the *Procedure return condition* box has been checked, and is used to enable the device ID automatic input function. This function enters the Device ID of the test record automatically. The value is given by *device\_id* which is a parameter of the response created by Start Procedure and Iteration Procedure.

**Repeat Stop Condition**

This area is used to set the repeat measurement stop condition. The following stop conditions are available.

- Counter Reaching to: *limit*  
The repeat measurement will be stopped if  $Count \geq limit$ . *Reset counter before repeating* box must be checked to set  $Count=0$  when the measurement is started.
- Procedure return condition  
The repeat measurement will be stopped if the response shows *status* = True.

Figure 1-4

**Procedure Response Expected by EasyEXPERT**

- Response (input to EasyEXPERT):

XML format data

```
<Response>  
<Break>status</Break>  
<Target>device_id</Target>  
</Response>
```

status: True (break) or False  
device\_id: Device ID

- Example:

```
<Response>  
<Break>False</Break>  
<Target>waf1a:3 1</Target>  
</Response>
```

## Thermo-trigger

This area is used to set the thermo-trigger function using the N1265A ultra high current expander/fixture.

### Enable

**Thermo-trigger** Enables/disables the thermo-trigger function.

### N1265A

### Thermo-trigger

**List...** Opens the Thermo-trigger List dialog box used to set the thermo-trigger function. See “Thermo-trigger List” on page 1-53.

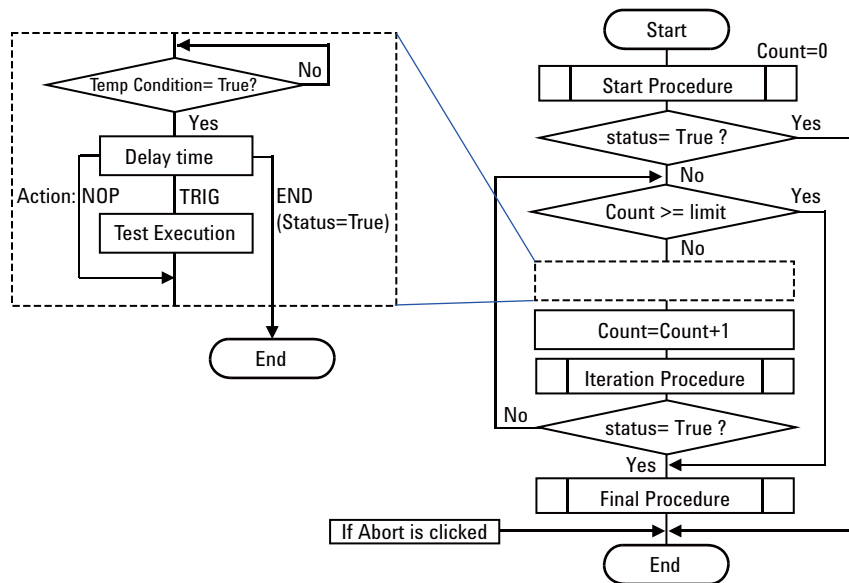
If the thermo-trigger function is enabled, the repeat measurement execution flow is changed as shown in Figure 1-5. The following steps are added.

1. After *Count* is judged, temperature (*Temp*) is measured.
2. If the measured temperature is in the range of the temperature condition (*Center* and *Radius*), the action (*Action*) is executed after the delay time (*Delay*) elapses. Or else, the temperature measurement and this judgement are repeated.

Where, the terms in ( ) are the setup items on the Thermo-trigger List dialog box.

Figure 1-5

### Repeat Measurement Execution Flow with Thermo-trigger Function





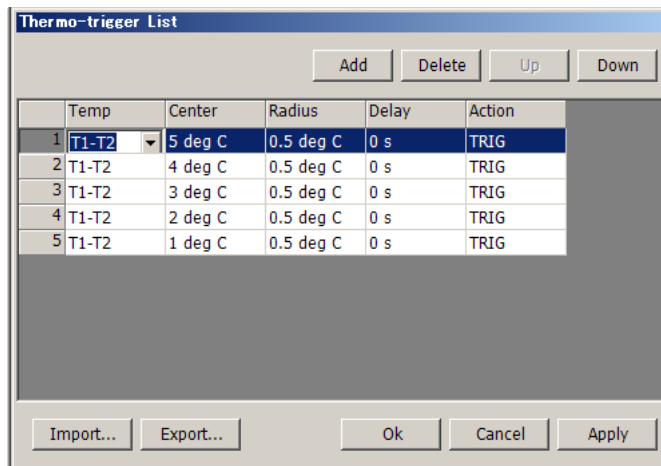
## Thermo-trigger List

This dialog box is opened by clicking the N1265A Thermo-trigger List button, and is used to set the Thermo-trigger function which is defined by listing multiple sets of the temperature condition and action executed when the condition is satisfied. A setup in a row of the list is used for one loop sequence of the repeat measurement. The top row is for the first sequence of the repeat measurement, the second row is for the second sequence, and so on. If the repeat measurement is repeated over the number of rows set in the list, the list is used from the top row again. This dialog box can have a maximum of 1001 rows.

And also, in this dialog box, you can import any list from an external file and export the list to an external file, in the CSV or XMLSS form.

Figure 1-6

Thermo-trigger List dialog box



- N (index)** Select an entry that you wish to edit during the editing. During the repeat measurement execution, this button indicates the current-waiting entry.
- Temp** Selects the temperature used in the condition from the following alternatives:
- T1: Temperature value measured by the thermometer channel 1.
  - T2: Temperature value measured by the thermometer channel 2.
  - T1-T2: Difference between T1 and T2.
- Center** Specifies the center value of the temperature range in deg C. Range: -50 to 300. Resolution: 0.5.

## Main GUI

### Repeat Measurement Setup

<b>Radius</b>	Specifies the radius value of the temperature range in deg C. Range: 0.5 to 50. Resolution: 0.5.
<b>Delay</b>	Specifies the delay time from the temperature condition satisfaction to starting of the action specified by the Action field, in seconds. Range: 0 to 3600, Resolution: 1.
<b>Action</b>	Selects an action executed if the measured temperature is in the specified range, from the following alternatives: <b>TRIG:</b> Does the single test execution after the specified delay time, and then goes to the next operation. This is useful to do the single test execution in the specified temperature range. <b>NOP:</b> No operation (single test execution is skipped). Waits for the delay time, and goes to the next operation. This is useful to wait in the specified temperature range. <b>END:</b> Sets the <i>status</i> to True, and exits the repeat measurement after the delay time. This is useful to exit the repeat measurement in the specified temperature range.
<b>Add</b>	Adds a new entry for the thermo-trigger condition and action above the selected entry.
<b>Delete</b>	Deletes the selected entry.
<b>Up/Down</b>	Moves the selected entry upward or downward.
<b>OK</b>	Applies the setup changes and closes this dialog box.
<b>Cancel</b>	Cancel the setup changes and closes this dialog box.
<b>Apply</b>	Applies the setup changes.

- Import...** Opens the Open File dialog box. Imports the specified file (.csv or .xml).
- Export...** Opens the Save File dialog box. Specify the file format in this dialog box.

**CVS:**

Exports the current settings as the specified name in the CSV format that can be read by a spread sheet software. The data is stored with the same order as the settings on the screen.

**XML Style Sheet:**

Exports the current settings as the specified name in the XML spread sheet format that can be read by a spread sheet software. The tags are as follows:

```
<Thermo-trigger List>  
  <Temp>A</Temp>  
  <Center>B</Center>  
  <Radius>C</Radius>  
  <Delay>D</Delay>  
  <Action>E</Action>  
</Thermo-trigger List>
```

Where,

A: Temp field setting, T1, T2, or T1-T2, characters

B: Center field setting, numeric value

C: Radius field setting, numeric value

D: Delay field setting, numeric value

E: Action field setting, TRIG, NOP, or END, characters

## Organize Preset Group

This dialog box is opened by selecting the Organize Preset Group function of the main screen, the Quick Test setup screen, or the Switching Matrix Operation Panel, and is used to organize the preset group.

<b>Preset Group List</b>	Lists the preset group saved in the workspace. The following buttons are available.
<b>Add</b>	Opens the Add Preset Group dialog box. Creates a new preset group and saves it as the specified name.
<b>Rename</b>	Opens the Rename Preset Group dialog box. You can change the name of the selected preset group.
<b>Duplicate</b>	Opens the Duplicate Preset Group dialog box. Creates a copy of the selected preset group and saves it as the specified name.
<b>Delete</b>	You can delete the selected preset group.
<b>Import</b>	Only for the quick test setup. Opens the Preset Group Import dialog box. Imports the specified preset group.
<b>Export</b>	Only for the quick test setup. Opens the Preset Group Export dialog box. Exports the selected preset group as the specified name.
<b>Copy</b>	Sets the operation of the >> button and the << button. They work as Copy when this box is checked, and as Move when this box is unchecked.
<b>&gt;&gt;</b>	Copies or moves the preset setup selected in the left Preset Group box to the right Preset Group box.
<b>&lt;&lt;</b>	Copies or moves the preset setup selected in the right Preset Group box to the left Preset Group box.
<b>Close</b>	Closes the Organize Preset Group dialog box.

**Preset Group**

Lists the preset setup saved in the preset group. The following buttons are available.

<b>Preset Group</b>	Selects the preset group to open.
<b>Setup Name</b>	Lists all preset setup saved in the preset group.
<b>Select All</b>	Selects all preset setup.
<b>Unselect All</b>	Unselects all preset setup.
<b>Rename</b>	You can change the name of the selected preset setup.
<b>Duplicate</b>	Creates a copy of the selected preset setup and places it at the bottom of the test setup list.
<b>Delete</b>	You can delete the selected preset setup.

## Calibration

The Calibration window is opened by clicking the Calibration button, and is used to perform the self-calibration, SMU current offset cancel, CMU data correction, and so on.

- “Module Self Calibration”
- “SMU Zero Cancel”
- “CMU Calibration”

### Module Self Calibration

The Module Self Calibration tab screen is used to perform the self-calibration of modules. Check the left check box of Name to select the module for self-calibration. Open the measurement terminals at the cable end before performing the self-calibration.

#### Module Self Calibration

Displays the following values.

<b>Name</b>	Name of module.
<b>Full Range</b>	Full range calibration On or Off. Only for SMU+ASU.
<b>Status</b>	Calibration status. Pass or fail.
<b>Notes</b>	Error information when calibration failed.

#### Select All

Selects all modules for calibration. This checks the check box for all modules.

#### Unselect All

Unselects all modules. This unchecks the check box for all modules.

#### Start Calibration

Starts calibration for the selected modules.

#### Abort

Aborts calibration.

#### Full Range Calibration

Only for the B1500A. Check this box to enable the 1 pA range calibration of SMU+ASU.

If you do not use the 1 pA range, uncheck this box. The B1500A does not perform the 1 pA range offset data measurement and calibration.

**Enable Auto Calibration**

Enables or disables the auto-calibration capability. If this box has been checked (function ON), and the B1500 automatically starts calibration for all modules every 30 minutes if the output switches of all modules are off for 30 minutes. If auto-calibration is enabled, do not forget to open the measurement terminals or disconnect the device under test from the terminals after measurement.

**SMU Zero Cancel**

The SMU Zero Cancel tab screen is used to perform the SMU path offset current measurement, and set the offset cancel function. Check the left check box of Name to select the effective module. Open the measurement terminals at the cable end before performing the offset measurement. This function is available for the following modules. See also “Current Offset Cancel” on page 4-43.

- SMU+ASU
- HRSMU
- MPSMU
- HPSMU
- HVSMU

**SMU Zero Cancel**

Displays the following values.

<b>Name</b>	Name of module
<b>Full Range</b>	SMU+ASU full range measurement On or Off.
<b>1nA</b>	Offset current measurement data in the 1 nA range. Available for SMU+ASU, HRSMU, MPSMU, HPSMU, and HVSMU.
<b>100pA</b>	Offset current measurement data in the 100 pA range. Available for SMU+ASU and HRSMU.
<b>10pA</b>	Offset current measurement data in the 10 pA range. Available for SMU+ASU and HRSMU.
<b>1pA</b>	Offset current measurement data in the 1 pA range. Available for SMU+ASU only.

**Full Range**

Enables (On) or disables (Off) the 1 pA range of the selected SMU+ASU.

**Integration Time**

Sets the integration time for the offset current measurement of the selected SMU path. 1 PLC (power line cycle) to 100 PLC, initial setting 16 PLC.

- Select All** Selects all modules. This checks the check box for all modules.
- Unselect All** Unselects all modules. This unchecks the check box for all modules.
- Measure...** Performs the SMU path offset current measurement for the effective modules.

## CMU Calibration

The CMU calibration tab screen is used to perform the measurement data correction of the multi frequency capacitance measurement unit (MFCMU). For the easy way, perform the phase compensation and the open correction at least.

See also “Error Correction” on page 4-25.

---

### NOTE

For a more accurate measurement, perform correction data measurement at the measurement frequency before starting the capacitance measurement.

If the measurement frequency is not included in the list of default frequencies below, click the Advanced Options... button and set the measurement frequency on the Frequency area of the Advanced Options for CMU Calibration window.

Default frequencies:

1 k, 2 k, 5 k, 10 k, 20 k, 50 k, 100 k, 200 k, 500 k, 1 M, 1.2 M, 1.5 M, 2 M, 2.5 M, 2.7 M, 3 M, 3.2 M, 3.5 M, 3.7 M, 4 M, 4.2 M, 4.5 M, 5 MHz

---

- Measure...** Opens a dialog box used to perform correction/compensation data measurement. To perform the measurement, follow the dialog box.

The correction/compensation measurement data is saved and managed by the workspace. And the last measured correction/compensation data will be loaded when the workspace will be opened next time.

- Advanced Options...** Opens the “Advanced Options for CMU Calibration (page 1-62)” dialog box used to set the frequencies for the correction data measurement and the reference values of the working standard.

- Phase Compensation** This check box is effective after the phase compensation data is measured, and is used to enable or disable the phase compensation. Uncheck this box to disable the phase compensation.

To measure the compensation data, open the measurement terminals at the end of the device side and click the Measure... button.



If you extend the measurement cables to the device end by using the four terminal pairs method, do not forget to connect following terminals together at the device end.

- the high potential (Hp) terminal and the high current (Hc) terminal
- the low potential (Lp) terminal and the low current (Lc) terminal

### **Open Correction**

This check box is effective after the open correction data is measured, and is used to enable or disable the open correction. Uncheck this box to disable the correction.

To measure the correction data, connect the open standard that has the calibration value or reference value, or open the measurement terminals at the end of the device side. Then, click the Measure... button.

If you extend the measurement cables to the device end by using the four terminal pairs method, do not forget to connect following terminals together at the device end.

- the high potential (Hp) terminal and the high current (Hc) terminal
- the low potential (Lp) terminal and the low current (Lc) terminal

If the standard is used, click the Advanced Options... button and enter the reference value to the OPEN Reference entry fields. If you do not use the standard, enter 0 to the fields.

### **Short Correction**

This check box is effective after the short correction data is measured, and is used to enable or disable the short correction. Uncheck this box to disable the correction.

To measure the correction data, connect the short standard that has the calibration value or reference value, or directly connect the measurement terminals at the end of the device side. Then, click the Measure... button.

If the standard is used, click the Advanced Options... button and enter the reference value to the SHORT Reference entry fields. If you do not use the standard, enter 0 to the fields.

### **Load Correction**

This check box is effective after the load correction data is measured, and is used to enable or disable the load correction. Uncheck this box to disable the correction.

To measure the correction data, connect the load standard that has the calibration value or reference value. Then, click the Measure... button.

Before performing the load correction data measurement, click the Advanced Options... button and enter the reference value to the LOAD Reference entry fields.

## Advanced Options for CMU Calibration

This dialog box is opened by clicking the Advanced Options... button, and is used to set the information required to measure the open/short/load correction data and the phase compensation data. This dialog box provides the following action button.

- OK** Applies the setup changes and closes this dialog box.
- Cancel** Cancel the setup changes and closes this dialog box.

### Frequency

You can select one of the following radio buttons to select the measurement frequency setup mode.

- Use factory specified frequencies (Default)

Usually, select this mode. 23 points are automatically set. They are 1 k, 2 k, 5 k, 10 k, 20 k, 50 k, 100 k, 200 k, 500 k, 1 M, 1.2 M, 1.5 M, 2 M, 2.5 M, 2.7 M, 3 M, 3.2 M, 3.5 M, 3.7 M, 4 M, 4.2 M, 4.5 M, and 5 MHz.

- Specify list of calibration frequencies

Select this mode when you want to set the frequencies independently. The frequency must be 1 kHz to 5 MHz. The number of frequencies must be 1 to 101.

Click Add to open a dialog box, and enter the value.

For the unnecessary value, select the value and click Delete.

- Specify frequencies by range

Select this mode when you want to set the frequencies sequentially and automatically.

Specify the start frequency, the stop frequency, the number of steps, and the scale LINEAR/LOG. The start frequency must be less than the Stop value and at least 1 kHz. The stop frequency must be more than the Start value and no more than 5 MHz. The number of steps must be 2 to 1001.

---

### NOTE

If the device measurement frequency is not equal to the correction data measurement frequency, calculation will be performed automatically to get the correction data for the device measurement frequency, and the calculated correction data will be used for the data correction of the measurement data.

---

**Integration Time** This area defines the integration time used for measuring phase compensation data or open/short/load correction data. The number of averaging samples (Mode=AUTO) or the averaging time (Mode=PLC) is set.

**Mode** A/D converter operation mode, AUTO or PLC

**Factor** Factor for the initial value. See “Mode, Factor” on page 4-20.

**Reference  
Standard Values**

The following entry fields are available to set the calibration values or the reference values of the open/short/load standard. If you use the standard, enter the values to the following entry fields.

- OPEN Reference: Capacitance and Conductance  
If you do not use the reference, enter 0 in the both fields.
- SHORT Reference: Inductance and Resistance  
If you do not use the reference, enter 0 in the both fields.
- LOAD Reference: Inductance and Resistance

## Configuration

The Configuration window is opened by clicking the Configuration button, and is used to display the system information and event log, as well as to perform diagnosis and self-test.

- “Main Frame”
- “Modules”
- “ASU”
- “Switching Matrix”
- “SMU/PG Selector”
- “Module Selector”
- “Dual HCSMU Combination”
- “UHC Expander / Fixture”
- “HVSMU Current Expander”
- “UHV Expander”
- “SMU Output Setting Limits”
- “Event Log”
- “Extended Configuration”

## Main Frame

The Main Frame tab screen is used to display the system information and perform diagnosis.

<b>Model Identification</b>	Displays the instrument ID. B1500A or B1505A.
<b>Line Frequency</b>	Sets the power line frequency, 50 Hz or 60 Hz.
<b>EasyEXPERT Rev.</b>	Displays the revision number of the EasyEXPERT.
<b>Firmware Rev.</b>	Displays the revision number of the B1500 firmware.
<b>HOST ID</b>	Displays the host id of the B1500 internal computer.

**Main Frame  
Diagnosis**

This area lists Item and Status.

**Item** Name of diagnosis item. To select, check the left check box.  
**Status** Diagnosis status. Pass or fail.

**Select All**

Selects all items for diagnosis. This checks the box for all items.

**Unselect All**

Unselects all items. This unchecks the box for all items.

**Start Diagnosis**

Starts diagnosis for the selected items. To perform diagnosis, specify the items to perform and click this button. A dialog box appears. Follow the dialog box to perform the diagnosis. Some items need your judgement, pass or fail. Also you will need to connect an adapter, press keys, check LEDs, and so on.

**Modules**

The Modules tab screen is used to perform self-test of modules.

**Slot Configuration**

This area lists Slot, Module Type, Name, and Status.

**Slot** Slot number where the module is installed. To select, check the left check box.  
**Module Type** Module model number  
**Name** Module name. See Table 1-2.  
**Status** Self-test status, Passed, Failed, Recovered, or ---- (self-test has not been performed)  
**SCUU** Specifies if the module is connected to the SCUU.  
**Notes** Error information when the module self-test failed.

**Select All**

Selects all modules for self-test. This checks the check box for all modules.

**Unselect All**

Unselects all modules. This unchecks the box for all modules.

**Start Self Test**

Starts self-test for the selected modules.

**Recover Module**

This button is for service use only. Makes the failed modules respond again. This button does not solve the problem that causes the Failed status.

**Abort**

Aborts self-test.

**Status LED**

Enables (ON) or disables (OFF) the status indicator (LED) of the ASU (Atto Sense and Switch Unit), the SCUU (SMU CMU Unify Unit), and the module selector.

**Table 1-2**

**Module Name**

Module name	Measurement resource
GNDU+ADC	B1500 mainframe
SMU<N>:HP	HPSMU
SMU<N>:MP	MPSMU
SMU<N>:HR	HRSMU
SMU<N>:HR/AS	SMU+ASU
SMU<N>:HC	HCSMU
SMU<N>:DHC	Dual HCSMU
SMU<N>:MC	MCSMU
SMU<N>:HV	HVSMU
SMU<N>:UHC500	UHCU (standard)
SMU<N>:UHC1500	UHCU (with the option N1265A-015)
SMU<N>:HVMC	HVMCU
SMU<N>:UHV	UHVU

Where, <N> is a channel number.

**ASU**

Only for the B1500A. The ASU tab screen is available if the B1500A installs the SMU calibrated with the ASU (Atto Sense/Switch Unit) regardless of the ASU connection status, and shows the ASU information.

**Slot**

Slot number

**Module Type**

Type of module

**Name**

Name of module

**ASU Serial Number**

Serial number of the usable ASU. This cell shows the serial number even if the ASU is disconnected.

If an incorrect ASU is connected to the SMU, this cell displays an error message instead of the serial number. Error message is also displayed if an ASU is connected to the SMU which is not available for using ASU.

**ASU I/O Path**

ASU switching operation, SMU or AUX

Value	Input-to-output connection and description
SMU	ASU always makes the path from the ASU Force input to the ASU output.
AUX	ASU performs automatic switching in every test. The ASU normally makes the path from the AUX In input to the ASU output and makes the path from the SMU input to the ASU output for the test using the SMU.

**Switching Matrix**

Not available for the B1505A. The Switching Matrix tab screen is used to confirm the GPIB connection with Agilent B2200A/B2201A/E5250A switching matrix, and defines the connections between the measurement modules and the switching matrix input ports.

**Switching Matrix Type**

Displays the type of the switching matrix. One of the following.

- B2200A Femto Leakage Switch Mainframe
- B2201A 14ch Low Leakage Switch Mainframe
- E5250A Low Leakage Switch Mainframe
- No Switching Matrix (if switching matrix is not connected)

**GPIB Address**

Specifies the GPIB address of the switching matrix. 1 to 30.

**Poll**

Confirms if the switching matrix with the specified GPIB Address is connected to the B1500A.

**Start Self Test**

This button starts the selftest of the switching matrix.

**SWM IN**

SWM IN $n$  means the switching matrix input port  $n$ . where,  $n$  is an integer 1 to 10 for the E5250A or an integer 1 to 14 for the B2200A/B2201A.

## Main GUI Configuration

Enter the label used to specify the input port to the right entry field. The port name and the connected module name can be the label. Or select one from the pull-down menu. There must be no colon (:) in the label.

If the Kelvin check box is checked, the coupled even number port is grayed out.

If the CMU check box is checked, the SWM IN10 for the E5250A or the SWM IN14 for the B2200A/B2201A is grayed out.

### Kelvin

Check this box if this port is used for the Kelvin connection. Connect the switching matrix inputs to the SMU as follows.

- for the E5250A ( $m$ : integer 1 to 3)  
SMU INPUT  $2m-1$  to the SMU Force terminal  
SMU INPUT  $2m$  to the SMU Sense terminal
- for B2200A/B2201A ( $m$ : integer 1 to 6)  
Input  $2m-1$  to the SMU Force terminal  
Input  $2m$  to the SMU Sense terminal

### CMU

Check this check box if this port is connected to the CMU. Connect the switching matrix inputs to the CMU as follows.

- for the E5250A  
AUX INPUT CV1 to the CMU Hpot/Hcur terminals  
AUX INPUT CV2 to the CMU Lpot/Lcur terminals
- for B2200A/B2201A  
Input 13 to the CMU Hpot/Hcur terminals  
Input 14 to the CMU Lpot/Lcur terminals

To obtain accurate capacitance measurement result, perform capacitance compensation. See Agilent B1500A *User's Guide* for the capacitance compensation when the switching matrix is used.

Also see “Extended Configuration” on page 1-79.

### Extended Configuration

This button opens the “Extended Configuration” on page 1-79 dialog box.



## SMU/PG Selector

Not available for the B1505A. The SMU/PG Selector tab screen is used to specify the input connection and the channel connection status of the Agilent 16440A SMU/PGU selector (B1500A-A04). The selector can be controlled by the B1500A via the Digital I/O connector and the Agilent 16445A selector adapter.

Two selectors can be used. The first selector must be connected to the Digital I/O connector via the 16445A. And if the second selector is used, it must be connected to the first selector's Control Output connector. See "SMU/PG Selector" on page 4-49.

- Enable SMU/PG Selector** To use the selector, connect the selector to the B1500A, and check this check box.
- Selector CH1/CH2** CH1 and CH2 indicates the channels 1 and 2 of the first selector respectively.
- Selector CH3/CH4** CH3 and CH4 indicates the channels 1 and 2 of the second selector respectively.
- SMU Assignment** Specifies the SMU connected to the SMU input of the selector channel CH1, CH2, CH3, or CH4.
- Input/Output Path** Switching operation of the selector channel CH1, CH2, CH3, or CH4

Value	Input-to-output connection and description
Always SMU	Selector always makes the path from Input SMU to Output.
Normally PGU (AUX)	Selector performs automatic switching in every test. The selector normally makes the path from Input PGU to Output and makes the path from Input SMU to Output only for the test which uses the SMU connected to the Input SMU port.

The CH1 and CH3 also provides the PGU Open status which is set by using the Advanced window of the Measurement Setup.

## Module Selector

Only for the B1505A. This tab screen is used to specify the input connection and the default input-to-output path of the module selector. The selector can be controlled by the B1505A via the Digital I/O connector.

## Main GUI Configuration

<b>Enable Module Selector</b>	This check box is used to enable/disable the module selector. If this box is checked, the module selector can be used.						
<b>Input</b>	<p>Specifies the modules connected to the module selector's HVSMU, HCSMU, and HPSMU input ports respectively. For the port which no module is connected, blank the entry field.</p> <p><b>Enable Series Resistor (100 kohm) for HVSMU</b></p> <p>This check box is used to connect/disconnect the built-in series resistor (100 k<math>\Omega</math>) on the HVSMU connection path. If this box is checked, the series resistor is connected.</p>						
<b>Output</b>	<p>Specifies the module selector connection status in the normal state. The module selector makes the path to the output port from the input port specified by the Default field.</p> <p>When a measurement is performed, the module selector performs automatic switching in every test. The selector makes the path to the output port from the module used for the test. The connection status is indicated by the Status indicator mounted on the module selector. See Table 1-3 on page 1-71.</p>						
<b>Status</b>	<p>Shows one of the following status.</p> <table><tr><td><b>Not Detected</b></td><td>Module selector has not been detected yet.</td></tr><tr><td><b>Communication Established</b></td><td>Normal status. Module selector can be used.</td></tr><tr><td><b>Communication Lost</b></td><td>Once the module selector was detected. However the communication has been broken now.</td></tr></table>	<b>Not Detected</b>	Module selector has not been detected yet.	<b>Communication Established</b>	Normal status. Module selector can be used.	<b>Communication Lost</b>	Once the module selector was detected. However the communication has been broken now.
<b>Not Detected</b>	Module selector has not been detected yet.						
<b>Communication Established</b>	Normal status. Module selector can be used.						
<b>Communication Lost</b>	Once the module selector was detected. However the communication has been broken now.						
<b>Auto Detection</b>	This check box is used to enable/disable the module selector automatic detection. If this box is checked, the automatic detection of the module selector is performed when EasyEXPERT is launched.						
<b>Diagnosis</b>	The Start Diagnosis button is used to start the module selector diagnosis. And the Status shows the diagnosis result.						
<b>Apply</b>	This button applies the setting.						

**Table 1-3 Module Selector Status Indicator and Input-to-Output Connection Path**

Module used for the test	Status indicator	Input-to-output connection <sup>a</sup> of output terminals			
		High Sense	High Force	Low Sense	Low Force
HPSMU or MPSMU	HPSMU	HP/MPSMU Sense	HP/MPSMU Force	HCSMU Low Sense + GNDU Force + GNDU Sense	HCSMU Low Force
HCSMU or DHCSMU	HCSMU	HCSMU High Sense	HCSMU High Force		
HVSMU	HVSMU	HVSMU Force	Open		
HVSMU and built-in series resistor		HVSMU Force + series resistor			
b	Open	Open	Open		

- a. In the normal state, the module selector makes the path to the output port from the input port specified by the Default field. When a measurement is performed, the module selector performs automatic switching in every test. The selector makes the path to the output port from the module used for the test.
- b. This condition is when all Input fields HVSMU, HCSMU, and HPSMU are blank.

**CAUTION**

If dual HCSMU (DHCSMU) is used with the module selector, the maximum current must be  $\pm 30$  A to prevent the module selector from performance degradation and failure.

**Dual HCSMU Combination**

Only for the B1505A installed with two HCSMU modules. This screen is used to set the dual HCSMU operation.

**Enable**

This check box is used to enable/disable the dual HCSMU operation. If this box is checked, the dual HCSMU operation is enabled and two HCSMU modules can be a SMU which supports  $\pm 40$  A (pulse),  $\pm 2$  A (DC).

**Master HCSMU**

Specifies the HCSMU module used as the master channel of the dual HCSMU operation. The module must be connected to the Master port of the 16493S-020/021 dual HCSMU adapter.

**Slave HCSMU** Specifies the HCSMU module used as the slave channel of the dual HCSMU operation. The module must be connected to the Slave port of the 16493S-020/021 dual HCSMU adapter.

**Apply** This button applies the dual HCSMU operating condition. To apply the setup changes successfully, the B1505A must be in the idle state and the Standby output must be OFF.

## UHC Expander / Fixture

Only for the B1505A. This tab screen is used to set the N1265A ultra high current expander/fixture. The N1265A can be controlled by the B1505A via the Digital I/O connector.

**Enable Ultra High Current Unit** This check box is used to enable/disable the ultra high current unit (UHCU). If this box is checked, UHCU can be used.

**Control** Specifies the modules used for controlling the UHCU. For the port which no module is connected, blank the entry field.

### Voltage Control

**SMU** Specifies the module (MCSMU or HCSMU) connected to the V Control port of the N1265A. The SMU channel set to this field is assigned to the UHCU and is used to indicate the UHCU.

### Current Control

**SMU** Specifies the module (MCSMU or HCSMU) connected to the I Control port of the N1265A.

**Self Test** Effective only when the Enable Ultra High Current Unit check box is checked.

The Start button starts the self-test for the N1265A. Measurement terminals must be opened during the self-test. The Status shows the execution result.

**Self Calibration** Effective only when the Enable Ultra High Current Unit check box is checked.

The Start button starts the self-calibration for the N1265A. Measurement terminals must be opened during the self-test. The Status shows the execution result.

**Enable Selector** This check box is used to enable/disable the selector of the N1265A. If this box is checked, the selector can be used.

**Input** **UHCU** Displays the SMU channel assigned to the UHCU.

<b>HVSMU</b>	Specifies the HVSMU connected to the HVSMU port of the N1265A directly or via the N1266A current expander.
<b>SMU</b>	Specifies the module (MPSMU or HPSMU) connected to the SMU port of the N1265A.

For the port which no module is connected, blank the entry field.

## Output

### Enable Series Resistor (100 kohm) for HVSMU

This check box is used to connect/disconnect the built-in series resistor (100 kΩ) on the HVSMU direct path. If this box is checked, the series resistor is connected.

**Default** Specifies the selector input-to-output connection status (UHCU, HVSMU, SMU, GNDU, or blank) in the normal state. The selector makes the path to the output port from the input port specified by the Default field. If this field is blank, the output port is open.

When a measurement is performed, the selector performs automatic switching in every test. The selector makes the path to the output port from the measurement resource used for the test. The connection status is indicated by the Status indicator mounted on the N1265A. See Table 1-4.

## Enable Gate Control

This check box is used to enable/disable the Gate path of the N1265A. If this box is checked, the Gate path can be used.

## Gate Control

**SMU** Specifies the module (MCSMU or HCSMU) connected to the Gate port of the N1265A. For the port which no module is connected, blank the entry field. Built-in series resistor can be connected to the Gate path. To connect the series resistor, set the Resistor field.

**Resistor** Specifies the resistance of the built-in series resistors for the Gate path. If the series resistor is not used, select 0 ohm.

**Table 1-4 N1265A Status Indicator and Selector Connection Path**

Status indicator	Selector Output terminals <sup>a</sup>			
	Low Sense	Low Force	High Sense	High Force
(off)	Open	Open	Open	Open
	GNDU Sense	GNDU Force	Open	Open
SMU	GNDU Sense	GNDU Force	SMU Sense	SMU Force
UHC	UHCU Low Sense	UHCU Low Force	UHCU High Sense	UHCU High Force
HVSMU	GNDU Sense	GNDU Force	HVSMU Force	Open
	GNDU Sense	GNDU Force	HVSMU Force + Series resistor	Open

a. In the normal state, the selector makes the path to the output port from the input port specified by the Default field. When a measurement is performed, the selector performs automatic switching in every test. The selector makes the path to the output port from the measurement resource used for the test.

**Thermometer**

- Channel 1** Enables/disables the thermometer channel 1.
- Channel 2** Enables/disables the thermometer channel 2.
- Interval** Specifies the interval of the temperature measurement. During executing any classic test, the temperature is not measured regardless of this setting.
- Format** Specifies the record format of the temperature data. The data is recorded by adding to the end of the Device ID string. Variables {0} for the channel 1 temperature, {1} for the channel 2 temperature, and {2} for the difference between {0} and {1} are available for setting the format.  
Initial setting: T1={0}, T2={1}, T1-T2={2} deg C

You can use the measurement temperature values as the repeat measurement condition. See “Repeat Measurement Setup” on page 1-50.

**Auto Detection** This check box is used to enable/disable the automatic detection of the N1265A. If this box is checked, the automatic detection is performed when EasyEXPERT is launched.

**Status** Shows one of the following status.

<b>Not Detected</b>	N1265A has not been detected yet.
<b>Communication Established</b>	Normal status. N1265A can be used.
<b>Communication Lost</b>	Once the N1265A was detected. However the communication has been broken now.

**Diagnosis** Effective when the Enable Ultra High Current Unit check box, the Enable Selector check box, or/and the Enable Gate Control check box is checked.

The Start button starts the diagnosis for the N1265A. Dialog box will be opened to navigate you the cable connection during the diagnosis. The Status shows the execution result.

**Apply button** This button applies the setting.

---

**NOTE**

**Before starting diagnosis**

If the N1266A expander is connected between the HVSMU and the N1265A, disconnect the N1266A. And connect the cable from the HVSMU Force terminal to the N1265A Selector Input HVSMU terminal.

---

## HVSMU Current Expander

Only for the B1505A. This tab screen is used to set the N1266A HVSMU current expander. The N1266A can be controlled by the B1505A via the Digital I/O connector.

**Enable High Voltage Medium Current Unit**

This check box is used to enable/disable the high voltage medium current unit (HVMCU). If this box is checked, HVMCU can be used.

**Input/Control**

Specifies the HVSMU and the modules used for controlling the HVMCU. For the port which no module is connected, blank the entry field.

**HVSMU** Specifies the HVSMU connected to the HVSMU input port of the N1266A.

**Voltage Control SMU** Specifies the module (MCSMU or HCSMU) connected to the V Control port of the N1266A. The SMU channel set to this field is assigned to the HVMCU and is used to indicate the HVMCU.

**Current Control**

**SMU** Specifies the module (MCSMU or HCSMU) connected to the I Control port of the N1266A.

**Self Test**

Effective only when the Enable High Voltage Medium Current Unit check box is checked.

The Start button starts the self-test for the N1266A. Measurement terminals must be opened during the self-test. The Status shows the execution result.

**Output**

**Enable Series Resistor (100 kohm) for HVSMU**

This check box is used to connect/disconnect the built-in series resistor (100 k $\Omega$ ) on the HVSMU direct path. If this box is checked, the series resistor is connected.

**Default** Specifies the input-to-output connection status (HVSMU, HVMCU, or blank) in the normal state. The N1266A makes the path to the output port from the input port specified by the Default field. If this field is a blank, the High output port is open.

When a measurement is performed, the N1266A performs automatic switching in every test. The N1266A makes the path to the output port from the measurement resource used for the test.

The connection status is indicated by the Status indicator mounted on the N1266A. See Table 1-5.

**Auto Detection**

This check box is used to enable/disable the N1266A automatic detection. If this box is checked, the automatic detection is performed when EasyEXPERT is launched.

**Status**

Shows one of the following status.

**Not Detected** N1266A has not been detected yet.

**Communication Established** Normal status. N1266A can be used.

**Communication Lost** Once the N1266A was detected. However the communication has been broken now.

**Apply button**

This button applies the setting.



**Table 1-5 N1266A Status Indicator and Input-to-Output Connection Path**

Status indicator	Input-to-output connection <sup>a</sup> of output terminals	
	Low	High
(off)	GNDU Force and Sense	Open
HVMCU		HVMCU High
HVSMU		HVSMU Force
		HVSMU Force + Series resistor

a. In the normal state, the N1266A makes the path to the output port from the input port specified by the Default field. When a measurement is performed, the N1266A performs automatic switching in every test. The N1266A makes the path to the output port from the measurement resource used for the test.

## UHV Expander

Only for the B1505A. This tab screen is used to set the N1268A ultra high voltage expander. The N1268A can be controlled by the B1505A via the Digital I/O connector.

### Enable Ultra High Voltage Unit

This check box is used to enable/disable the ultra high voltage unit (UHVU). If this box is checked, UHVU can be used.

### Control

Specifies the modules used for controlling the UHVU. For the port which no module is connected, blank the entry field.

#### Voltage Control

##### SMU

Specifies the module (MCSMU or HCSMU) connected to the V Control port of the N1268A. The SMU channel set to this field is assigned to the UHVU and is used to indicate the UHVU.

#### Current Control

##### SMU

Specifies the module (MCSMU) connected to the I Control port of the N1268A.

### Auto Detection

This check box is used to enable/disable the N1268A automatic detection. If this box is checked, the automatic detection is performed when EasyEXPERT is launched.

## Main GUI Configuration

<b>Status</b>	Shows one of the following status. <b>Not Detected</b> N1268A has not been detected yet. <b>Communication Established</b> Normal status. N1268A can be used. <b>Communication Lost</b> Once the N1268A was detected. However the communication has been broken now.
<b>Diagnosis</b>	Effective when the Enable Ultra High Voltage Unit check box is checked. The Start button starts the diagnosis for the N1268A. Dialog box will be opened to navigate you the cable connection during the diagnosis. The Status shows the execution result.
<b>Apply button</b>	This button applies the setting.

### SMU Output Setting Limits

Only for the B1505A. This screen is used to set the upper limit of the SMU output value and compliance value.

<b>Voltage Setting Limit</b>	Sets the upper limit of the voltage output value and compliance value. Effective value: 200 V to 10 kV, in 100 V step.
<b>Current Setting Limit</b>	Sets the upper limit of the current output value and compliance value. Effective value: 1 A to 1500 A, in 1 A step.
<b>Apply</b>	This button applies the setting.

### Event Log

The event log tab screen displays the event log.

<b>Date/Time</b>	Displays date and time of the event.
<b>Message</b>	Event message
<b>ID</b>	Process ID of the event
<b>Process</b>	Process name
<b>User</b>	User name. Windows login name.

**Host** Host name of the B1500

**Refresh** Refreshes the event log.

## Extended Configuration

Not available for the B1505A. This dialog box opens by clicking the Extended Configuration button in the Configuration window's Switching Matrix screen and closes by clicking the Close button.

**Save as Hardware Profile** Switching matrix hardware configuration can be saved as the hardware profile.

**Save** Saves the present hardware profile.

The hardware profile can be made for each workspace individually.

## CMU Compensation

Selects the compensation mode of the capacitance measurement using the switching matrix. Click one of the radio buttons, No Compensation, Select, and User Compensation Data File. See Agilent B1500A *User's Guide* for the capacitance compensation when the switching matrix is used.

- No Compensation

Select this radio button if you do not need the capacitance compensation. The raw data without compensation is displayed and saved.

- B2200A/B2201A (standard)

The following setup is available for the B2200A/B2201A hardware profile.

- Select

Select this radio button to perform the capacitance compensation for the path from the B2200A/B2201A input ports to Agilent 16495F/G connector plate or Agilent B2220A probe card interface. The data after compensation is displayed and saved.

**DUT I/F** Specifies the model number of the path you use. The selections are the 16495F/G and B2220A.

**Cable** Specifies the model number of the cable from the B2200A/ B2201A output to the DUT I/F. The selections are the 16494A/B/C-001 (1.5 m) / 002 (3 m) / 005 (4 m).

This mode cannot perform the capacitance compensation for the cables to positioner/manipulator or the probe card.

- User Compensation Data File

## Main GUI Configuration

Select this radio button and specify the compensation data file to perform the capacitance compensation for the path from the B2200A/B2201A input ports to the end of positioner/manipulator/probe card. See Agilent B1500A *User's Guide* for creating the compensation data file. The data after compensation is displayed and saved.

- E5250A (option)

The following setup is available for the E5250A hardware profile.

- User Compensation Data File

Select this radio button and specify the compensation data file to perform the capacitance compensation for the path from the E5250A input ports to the end of positioner/manipulator/probe card. See Agilent B1500A *User's Guide* for creating the compensation data file. The data after compensation is displayed and saved.

<b>Clear</b>	Clears the User Compensation Data File field.
<b>Triaxial Cable</b>	Enter the length of the 16494A triaxial cable connected between the E5250A outputs to the connector plate.
<b>User Triaxial Cable</b>	Enter the length of the triaxial cable between the connector plate and the DUT. Enter 0 if the triaxial cable is not used.
<b>User Coaxial Cable</b>	Enter the length of the coaxial cable between the connector plate and the DUT. Enter 0 if the coaxial cable is not used.

## Switching Matrix Operation Panel

Not available for the B1505A. The Switching Matrix Operation Panel is opened by clicking the Switching Matrix button, and is used to create the switch setup for Agilent B2200A/B2201A/E5250A switching matrix, send the setup to the switching matrix, and so on. The switch setup can be saved to the switch setup preset group. This group is different from the preset group for test setups.

### Switch Setup

This area is used to create the switching matrix switch setup. You can create the setup by clicking the Input-Output cross point. The switch setup can be sent to the switching matrix by clicking the Apply Switch Setup button.

Color of cross point	Switch setup
green	close
gray	open

<b>Setup Name</b>	Name of switch setup
<b>Clear All</b>	Sets all cross point open.
<b>Input</b>	Shows the switching matrix input port and the measurement resource connected to it. This information is set in the “Configuration”.
<b>Label</b>	Shows the label set to the measurement resource. The label is set in the application test definition or the classic test setup.
<b>ALL</b>	<p>Checking this box sets all of this line to close.</p> <p>Unchecking this box sets all of this line to open.</p> <p>This box is gray and have a check when a cross point in this line is set to close.</p>
<b>Output</b>	Output channels 1 to 12, 24, 36, or 48. Available numbers depend on the number of the matrix cards installed in the switching matrix. The field below the channel number is used to enter the label for the output channel.

**Export to Current My Favorite Group** This button converts the switch setup to the switching matrix setup data format of the classic test, and saves it to the present My Favorite Setup group.

Main GUI  
Switching Matrix Operation Panel

<b>Apply Switch Setup</b>	This button sends the switch setup to the switching matrix.
<b>Read Switch Setup</b>	This button reads the switch setup from the switching matrix.
<b>Open All</b>	This button opens all relays on the switching matrix.
<b>Preset List</b>	<p>The switch setups you create <b>MUST</b> be saved to the preset group to reuse them.</p> <p>Click the Preset List button to display the menu that provides the following functions. Select the preset group by using the field below the button. The switch setups can be recalled by using the area below the field.</p> <p><b>Cancel</b> Closes the menu.</p> <p><b>Preset Group</b> Provides the following three functions.</p> <p><b>Add New Preset Group</b> Opens the Add Preset Group dialog box. Creates a new preset group and saves it as the specified name.</p> <p><b>Rename This Preset Group</b> Opens the Rename Preset Group dialog box. You can change the name of the selected preset group.</p> <p><b>Duplicate This Preset Group</b> Opens the Duplicate Preset Group dialog box. Creates a copy of the present preset group and saves it as the specified name.</p> <p><b>Delete This Preset Group</b> You can delete the selected preset group.</p> <p><b>Organize Preset Group...</b> Opens the Organize Preset Group dialog box used to organize the preset group. See “Organize Preset Group” on page 1-56.</p> <p><b>Rename</b> You can change the name of the selected switch setup.</p> <p><b>Duplicate</b> Creates a copy of the selected switch setup.</p> <p><b>Delete</b> Deletes the selected switch setup.</p> <p><b>Delete All</b> Deletes all switch setups in the present preset group.</p>
<b>Save</b>	The button saves the present switch setup into the present preset group as a new setup of the group.
<b>Recall</b>	The button recalls the selected switch setup. This means opening and displaying the definition of the selected switch setup.

## Standby Channel Definition

The Standby Channel Definition window is opened by clicking the Standby button, and is used to define the standby channels and set the standby mode. See “Standby Function” on page 4-38.

The standby channel is the channel for maintaining specified output when the standby mode is ON and in the non-measurement state. The standby channels will be in the standby state, while the other channels will be in the idle state which is 0 V output and 100  $\mu$ A compliance. When the standby mode is OFF and in the non-measurement state, all channels will be in the idle state.

### Standby Channel Control

This area provides the following button.

**Standby ON/OFF** Sets the standby mode ON or OFF.

### Standby Channel Definition

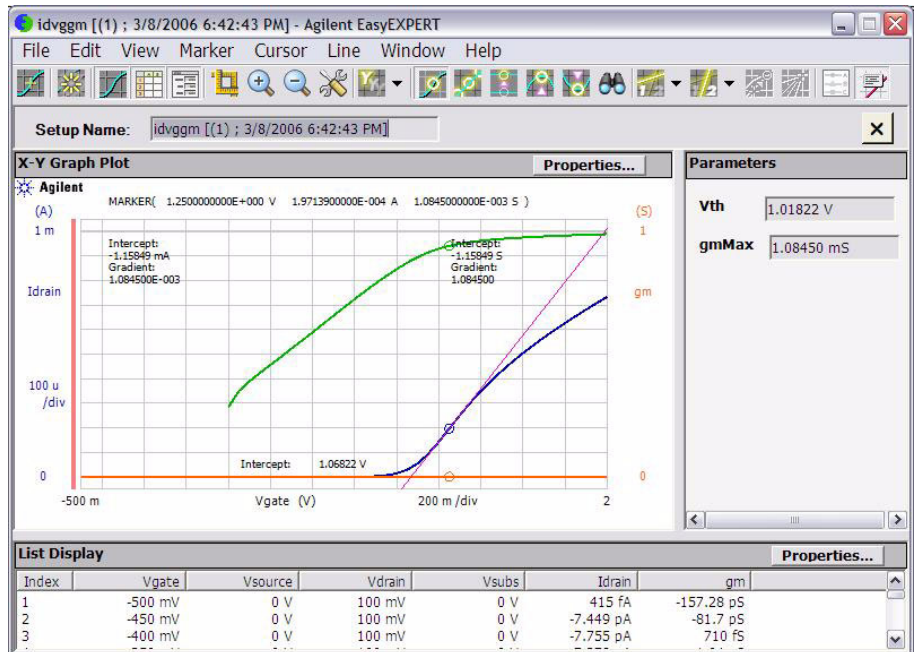
This area defines the standby channels. The standby channels will start to output in order, from the top to bottom, as listed in this area. The order of bias stop is bottom to top.

<b>Standby</b>	Check the check box to specify the standby channel.  The radio button is used to select the channel setup available for the Up/Down button.
<b>Unit</b>	SMU or UHVU used for the standby output. To specify the measurement resource connected to the selector (N1258A, N1259A-300, N1265A) or N1266A expander, the channel must be set to the Default output in the Configuration window.  Standby channel cannot be set for the SMU connected to the ASU which makes the path to the AUX port.
<b>Mode</b>	Output mode, V (voltage) or I (current)
<b>Source</b>	Standby channel output value, in V or A
<b>Compliance</b>	Compliance value, in A or V. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Up</b>	Moves the selected channel setup upward. To select a channel setup, use the radio button on the left.
<b>Down</b>	Moves the selected channel setup downward.

## Data Display

The Data Display window is opened by clicking the Data Display button, and is used to display measurement data and analyze the data.

- “Data Display GUI”
- “Display Setup”
- “Graph Properties”
- “List Display Properties”
- “Tool Bar”
- “Setup Summary”
- “Data Status”
- “Miscellaneous Operations”





---

**NOTE**

**About Display Mode**

The Data Display window provides two display mode Overwrite and Append.

In Overwrite mode, the “Display Data (page 1-17)” function opens a new Data Display window and displays data.

In Append mode, the “Display Data (page 1-17)” function creates a new display layer on the present Data Display window and displays data.

---

---

**NOTE**

**Updating Data Calculation**

When the classic test result is displayed, you can update the calculation results of the user function, analysis function, and auto analysis function. So use the Apply button. See “Function Setup (page 2-42)”.

---

---

**NOTE**

**Append Measurement**

After the Append measurement, you will see tabs between the tool bar and the graph plot area. The tabs are used to select the display layer for data display and analysis operation. The Data Display window can have maximum 10 display layers.

---

---

**NOTE**

**Multi Display Function**

In the Multi Display OFF status, the test result data will be always displayed on the singular Data Display window even if the test setup is changed.

In the Multi Display ON status, the test result data of the same test setup name will be displayed on the exclusive Data Display window and the test result data of the different test setup name will be displayed on the new Data Display window.

---

---

**NOTE**

The settings on the Display Setup are applied to the X-Y Graph on the Data Display window. The X-Y Graph can handle one X data on the X axis and up to eight Y data on the Y axis.

Define the X axis carefully when plotting Multi Channel I/V Sweep measurement results that may contain data measured by multiple VAR1 channels, as the result may contain multiple X data that can be selected for the X axis.

The Display Setup must be set properly to match the X axis data and the X data of the plot to be displayed.

---

## Data Display GUI

<b>Setup Name</b>	This field just displays the name of the test setup used to get this test record.
<b>X-Y Graph Plot</b>	This area displays the measurement result graph, traces, marker, cursor, and lines. The Properties button opens “Graph Properties (page 1-96)”.
<b>Parameters</b>	This area lists the user-defined output parameters if the output parameters are set.
<b>List Display</b>	This area displays the measurement result data list. The Properties button opens “List Display Properties (page 1-97)”.
<b>pin</b>	<p>The Data Display window can keep up to 10 display layers. And the earliest layer over 10 will be automatically deleted. This button is available when the window keeps at least two layers. If this button shows the standing pin, the layer will be kept. If this button shows the laid pin, the layer can be deleted.</p> <p>Double clicking the tab gives the same result as this button.</p>
<b>X</b>	This button closes the selected display layer.
<b>File menu</b>	Provides the following functions.
<b>Switch to Append Mode / Switch to Overwrite Mode</b>	Used to switch the display mode.
<b>Update Test Result</b>	Stores the information of the scale and the analysis tools.
<b>Delete Test Result</b>	Deletes the present display data or the selected display layer.
<b>Close This Display</b>	Closes the selected display layer.
<b>Close All Displays</b>	Closes all display layers.
<b>Save Image As...</b>	Opens the Save Image As... dialog box used to specify the file name of the graph image to save. See Figure 1-7. Available data formats are the BMP, EMF, GIF, and PNG.
<b>Page Setup...</b>	Opens the Page Setup dialog box used to set the print page.

**Print Preview**

Opens the Print Preview dialog box used to specify *Print Range/Data to print/Scaling* of the graph image to display. Clicking the Preview button opens the Print preview window. See Figure 1-8.

- Print Range sets Whole Data Display window or Graph Only.
- Data to print sets Current Data Display window only or All Data Display windows now opened.
- Scaling sets Adjust To the specified percent of the normal size or Fit To Page (automatic adjustment).

**Print...**

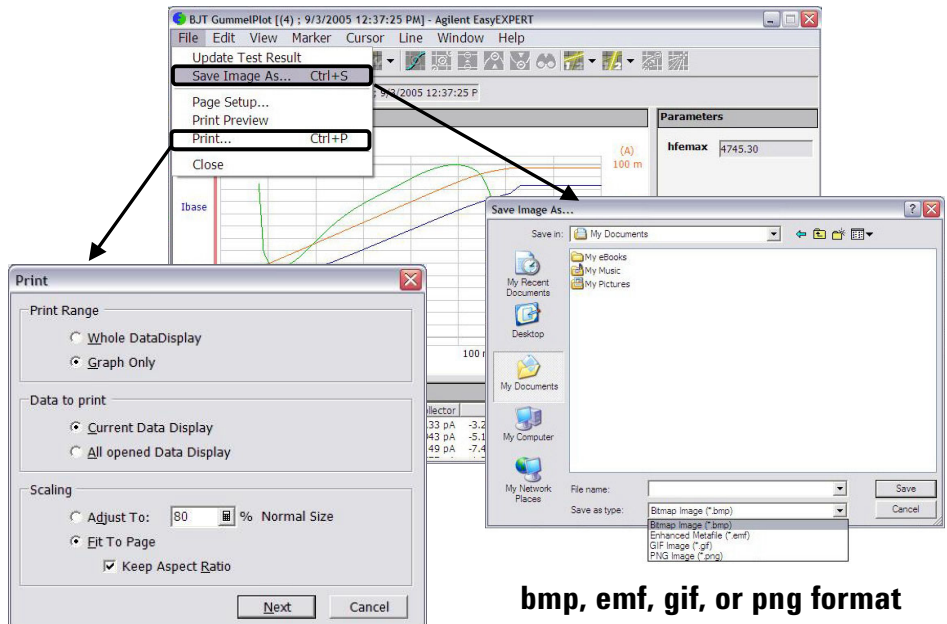
Opens the Print dialog box used to specify *Print Range/Data to print/Scaling* (see Print Preview) of the graph image to print. See Figure 1-7. Clicking the Next button opens another Print dialog box used to select the printer, print range, and number of copies, and start print.

**Close**

Closes the Data Display window.

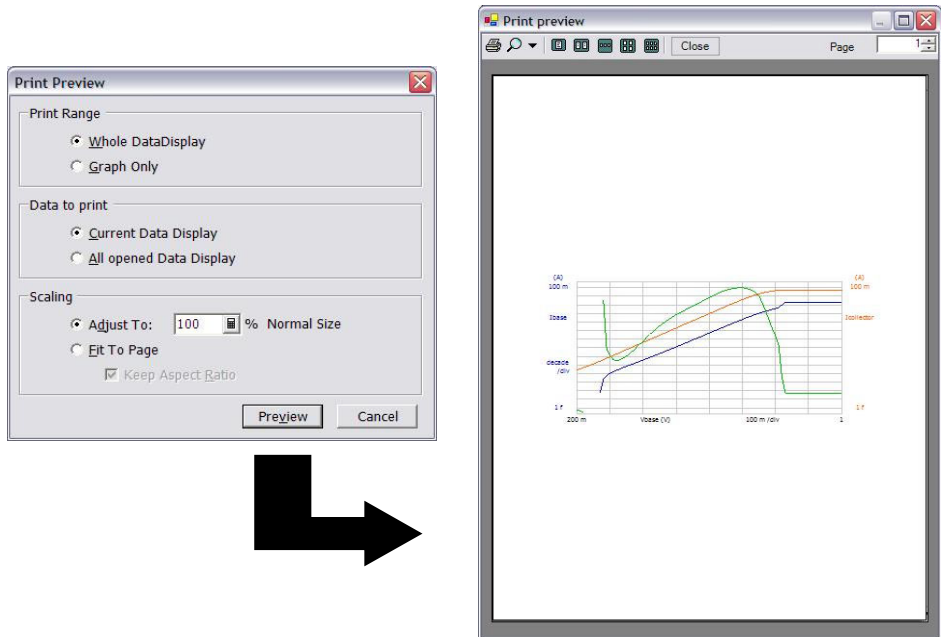
**Figure 1-7**

**Save Image As... and Print...**



**bmp, emf, gif, or png format**

Figure 1-8 Print Preview Example



## Edit menu

Provides the following functions.

### Copy

When the X-Y Graph Plot area is active, this function copies the graph image as BMP and EMF into the clipboard.

When the line cursor in the List Display area is active, this function copies the data list as CSV.

When the Parameters area is active, this function copies the parameter data as CSV.

### Copy Graph

Copies the graph image as BMP and EMF into the clipboard.

### Copy List

Copies the list data as CSV.

### Copy Parameters

Copies the parameter data as CSV.

### Graph Properties...

Opens “Graph Properties (page 1-96)”.

### List Display Properties...

Opens “List Display Properties (page 1-97)”.

Figure 1-9 shows the graph copy and paste example. Click the X-Y Graph Plot area on the Data Display window, select Edit > Copy menu, and paste it to WordPad. You will see the result as shown in this example.

Figure 1-9 Graph Copy and Paste

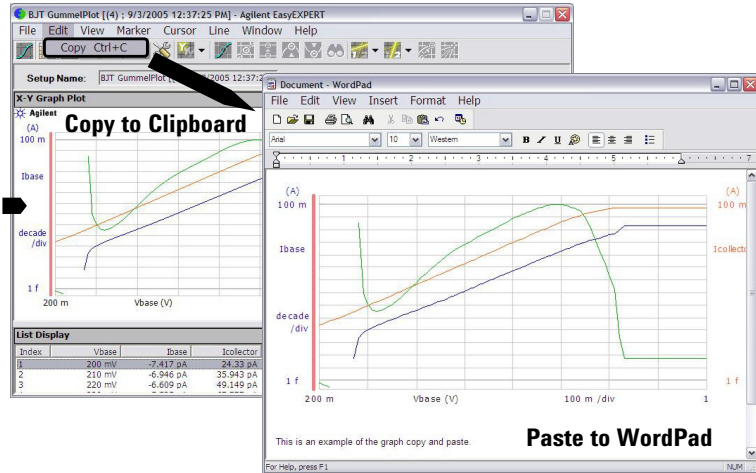
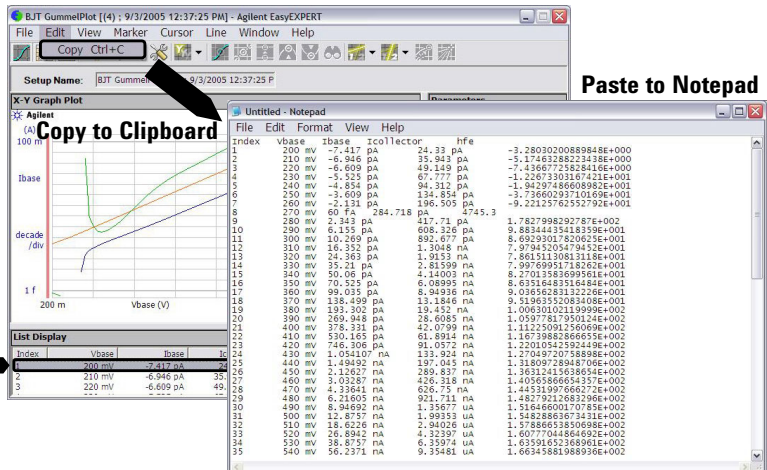


Figure 1-10 shows the list copy and paste example. Click a line of the List Display area on the Data Display window, select Edit > Copy menu, and paste it to Notepad. You will see the result as shown in this example. The data in Notepad is a CSV data (separated by a comma). Save it as a text file and open it by using a spreadsheet software. Each data (comma separated value) will be entered to a separate cell.

Figure 1-10 List Copy and Paste



## View menu

Provides the following functions.

<b>X-Y Graph</b>	Displays or hides the X-Y Graph Plot area.
<b>List Displays</b>	Displays or hides the List Display area.
<b>Parameters</b>	Displays or hides the Parameters area.
<b>Graph Legend ON/OFF</b>	Displays or hides the graph legend below the plot area.
<b>Line Information ON/OFF</b>	Displays or hides the line information (X-intercept, Y-intercept, gradient) in the plot area.
<b>Open Preview Window</b>	Opens “Preview (page 1-108)” and “Preview Settings (page 1-110)”.
<b>Auto Scale</b>	Changes graph scale to fit the trace in the graph automatically.
<b>Run Time Auto Scale</b>	Enables or disables the run time auto scaling. This function is set to independently for the X and Y axes.  If this function is enabled, the graph scale will be changed automatically to fit the trace in the graph during the measurement.
<b>Zoom In</b>	Zooms the data graph in the cursor. If the cursor is not displayed, zooms the data graph in the last cursor position or the graph center. This enlarges the trace in the graph.
<b>Zoom Out</b>	Zooms the data graph out the cursor. If the cursor is not displayed, zooms the data graph out the last cursor position or the graph center. This reduces the trace in the graph.
<b>Cancel Scaling</b>	Returns to the initial graph scale. This Display works only for the selected layer. All Displays works for the all display layers.
<b>Choose Active Y Axis</b>	Selects the Y-axis that is effective for the marker control and the auto scale.
<b>Display Setup...</b>	Opens “Display Setup (page 1-95)”.

If multiple layers are opened on the Data Display window, the scale change is applied to all layers. This means that the X-Y graph scale will be shared by all display layers. The auto scaling is performed as follows.

1. Graph scale is optimized for the data on the selected layer (top layer).
2. The graph scale is applied to the graph in all layers.

Note that auto scaling is performed independently for the left and right Y axes.

### Marker menu

Marker can move on the measurement data curve. The Marker menu provides the following six functions.

<b>Marker ON/OFF</b>	Enables or disables the marker.
<b>Interpolation ON/OFF</b>	Enables or disables the interpolation function of measurement data. You can read the interpolation data between two actual measurement points.
<b>Skip</b>	Moves the marker to the next measurement curve that is added by the VAR2 variable.
<b>Go To Maximum</b>	Moves the marker to the maximum value point.
<b>Go To Minimum</b>	Moves the marker to the minimum value point.
<b>Go To...</b>	Opens the Direct Marker/Cursor dialog box used to specify the coordinate values to move the marker.

### Cursor menu

Cursor can move anywhere on the graph. The Cursor menu provides the following three functions.

<b>Cursor ON/OFF</b>	Enables or disables the cursor.
<b>Go to Marker</b>	Moves the cursor to the marker position.
<b>Go to...</b>	Opens the Direct Marker/Cursor dialog box used to specify the coordinate values to move the cursor.

### Line menu

Provides the following five functions. When a line is enabled, gradient, X-intercept, and Y-intercept are displayed in the graph. See Figure 1-11.

<b>Line 1 ON/OFF</b>	Enables or disables line 1.
<b>Line 2 ON/OFF</b>	Enables or disables line 2.
<b>Line Mode</b>	Selects the line mode: normal, gradient, tangent, regression, or fix.
<b>Adjust Gradient</b>	Available when a gradient line is active. Checking this function enables the rotary knob and the mouse wheel to increase/decrease the gradient of the line. Unchecking this function, clicking the rotary knob, or clicking the mouse wheel disables this function.

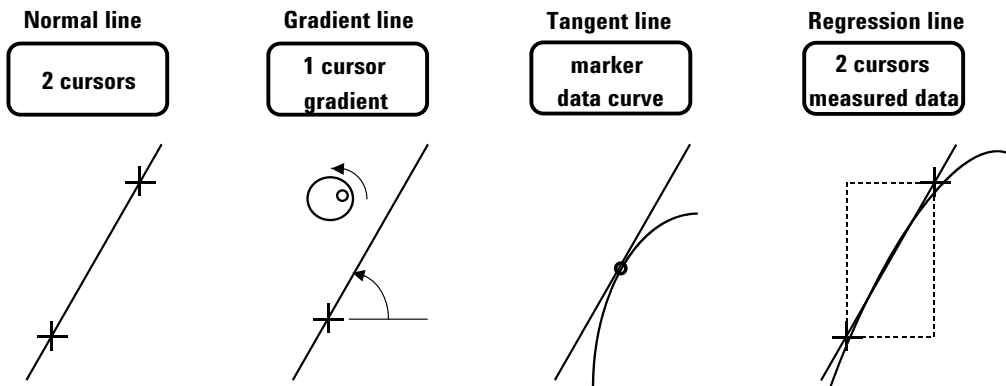
**Gradient...** Opens the Gradient Value dialog box used to specify the gradient value of the gradient line.

You can draw a line as shown below:

1. Set Line 1 or Line 2 on.
2. Select a Line Mode.

<b>Normal</b>	A line and two cursors appear. Move cursors to the points where the line passes through.
<b>Gradient</b>	A line and a cursor appear. Move cursor to the point where the line passes through. Enable Adjust Gradient function, then increase/decrease the gradient of the line by using the rotary knob or the mouse wheel.
<b>Tangent</b>	A line appears. Set the marker on. The tangent line appears on the marker point.
<b>Regression</b>	A line and two cursors appear. Move cursors for specifying the measurement data used for the regression calculation. The calculation is executed by using the data included in the square drawn by the extended cross cursor lines. For the subordinate sweep measurement data, specify the data curve by using the marker.
<b>Fix</b>	Fixes the line on the graph.

Figure 1-11 Line Mode





## Text menu

Up to 20 texts can be added to a graph. The following functions are provided.

**New** Opens the text editor for adding a new text.

**Hide All/ Show All** Hides or shows all texts.

**Delete All** Deletes all texts.

Clicking on a text will select the text. To select multiple texts, click the texts while holding down the Shift key on the keyboard.

The following functions are available for the selected text(s) via the Text menu, the Text icons, or the right-click menu.

**Edit Mode** Edits the selected text.

**Size** Sets the font size of the selected text(s). The available setting values are Largest, Large, Medium, Small, Smallest, and Fixed Point (6, 7, 8, 9, 10, 11, 12, 14, 16, 18, or 20). For settings other than Fixed Point, an appropriate size will automatically be applied. For Fixed Point, the specified font size will always be applied.

**Font** Sets the font of the selected text(s).

**Text Color** Sets the text color of the selected text(s). Click Customize to open the color palette.

**Background Color** Sets the background color of the selected text(s). Click Customize to open the color palette.

**Hide Frame/Show Frame** Hides or shows the frame for the selected text(s).

**Delete** Deletes the selected text(s).

## Pointer menu

Up to 30 pointers can be added to a graph. The following functions are provided.

<b>New</b>	This function is enabled when the marker is ON. Adds a new pointer at the position of the active marker.
<b>Hide All/ Show All</b>	Hides or shows all pointers.
<b>Show All Descriptions</b>	Shows all descriptions (pointer ID and X-Y coordinate values) of the pointers.
<b>Hide All Descriptions</b>	Hides all descriptions of the pointers.
<b>Delete All</b>	Deletes all pointers.

Clicking on a pointer will select the pointer. To select multiple pointers, click the pointers while holding down the Shift key on the keyboard.

The following functions are available for the selected pointer(s) via the Pointer menu, the Pointer icons, or the right-click menu.

<b>Shape</b>	Sets the shape of the selected pointer(s).
<b>Hide Description/Show Description</b>	Hides or shows the description of the selected pointer(s).
<b>Delete</b>	Deletes the selected pointer(s).

## Window menu

Provides the three functions below. The functions work for opened windows.

<b>Tiling</b>	Tiles all Data Display windows on the screen.
<b>Stacking</b>	Stacks all Data Display windows on the screen.
<b>Overlaying</b>	Overlays all Data Display windows on the screen. Then the position, window size, and visibility of the most front window are applied to all windows on the screen. The background of the X-Y Graph area is cleared except for the most back window.

## Help menu

<b>Agilent EasyEXPERT Help</b>	Opens the EasyEXPERT help window.
--------------------------------	-----------------------------------

## Display Setup

This setup screen is displayed by selecting View > Display Setup..., and is used to select the data displayed in the Data Display window.

### X-Y Graph

Selects the data to be displayed in the X-Y Graph Plot area. The X and Y1 to Y8 axes can be set.

<b>Name</b>	Specifies the parameter for the axis.
<b>Sharing</b>	Specifies the group for sharing the Y axis scale. Select from Group 1, Group 2, Group 3, Group 4, or None (no grouping).  The Scale, Min, and Max values are shared by the Y axes in the same group. The scale set by Run Time Auto Scale, Auto Scale, Zoom in, and/or Zoom out are also shared.  If another layer on the Data Display window contains an X-Y trace which has the same name for both its grouped Y parameter and its X parameter, the scaling will also be shared by this X-Y trace.
<b>Scale</b>	Scale of the axis, Linear or Log
<b>Min</b>	Minimum value of the axis
<b>Max</b>	Maximum value of the axis
<b>Add</b>	Adds a parameter (or axis).
<b>Delete</b>	Deletes the parameter selected with the radio button on the left.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

### List Display, Parameters

Selects the data displayed in the List Display area or the Parameters area. Up to 20 parameters can be set for each area.

<b>parameter</b>	Specifies the parameter for the data output.
<b>Add</b>	Adds a row for the parameter.
<b>Delete</b>	Deletes the parameter selected by the left radio button.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

## Graph Properties

This dialog box is opened by clicking the Properties... button in the X-Y Graph Plot area or by selecting Edit > Graph Properties..., and is used to set the graph plot area. Click the OK button to apply the setup and close this dialog box.

### Effective Area

This area provides the following check box.

**Logo ON**                      Displays the Agilent logo at the upper left of the graph area.

**Grid ON**                        Displays the grid on the graph.

#### Setup Summary

**ON**                                This is valid only for the I/V Sweep and Multi Channel I/V Sweep classic tests. Displays the channel setup summary at the bottom of the graph area. See “Setup Summary” on page 1-101.

### Graph Color

This area is used to set the color map for the graph plot area. You can change the color of the background, text, graph grid, and line/cursor. The color palette is opened by clicking the pattern of the item.

### Graph Trace Color

This area is used to set visibility, color, and thickness of the data traces Y1 to Y8 plotted on the graph. The color palette is opened by clicking the pattern of the item.

### Log Scale

This area is used to set the maximum number of decades to show the minor ticks on the log axis. If the number of graph decades is less than or equal to the entered value, the minor ticks are displayed. If it is greater than the entered value, the minor ticks are not displayed.

### Text Size

This area is used to set the font size for the graph. The available settings are Largest, Large, Medium, Small, Smallest, and Fixed. For settings other than Fixed, the appropriate size is automatically selected. For Fixed, a font size entry field is provided, and the specified font size is always applied.

**X/Y Axis Title**              Font size for the axis title

**Scale**                            Font size for the X and Y coordinate values

**Legend**                         Font size for the graph legend

## List Display Properties

This dialog box is opened by clicking the Properties... button in the List Display area or by selecting Edit > List Display Properties..., and is used to set the data display format in the List Display area. Click the OK button to apply the setup and close this dialog box.

### Display

Check the following check box to add the data status or the physical unit to data. Unchecking the box removes it.

**Data Status** Adds or removes the status code before data.

**Physical Unit** Adds or removes the physical unit after data.

### Number Format

Selects the data display format from the following formats.

- Engineering Format (Example: -1 . 2345 mA)  
Data display with arithmetic point, SI prefix, and unit.
- Scientific Notation (Example: -1 . 23456789012345E-003 A)  
Data display with arithmetic point, exponential part (E, +/- sign, and three-digit number), and unit.

## Tool Bar

The Data Display window provides the 22 icons on the tool bar. You will see the following icons from left to right in the Data Display window.

### Switch Display Mode

This icon switches the display mode.

In Overwrite mode, the “Display Data (page 1-17)” function opens a new Data Display window and displays data.

In Append mode, the “Display Data (page 1-17)” function creates a new display layer on the present Data Display window and displays data.

### Close All Displays

This icon closes all display layers.

### Show X-Y Graph

This icon displays or hides the X-Y Graph Plot area.

### Show List Display

This icon displays or hides the List Display area.

### Show Parameters

This icon displays or hides the Parameters area.

<b>Auto Scale</b>	<p>This icon will change the graph scale automatically to fit the trace in the graph.</p> <p>The right arrow button opens the menu for enabling or disabling the run time auto scaling. This function is set independently for the X and Y axes. If this function is enabled, the graph scale will be changed automatically to fit the trace in the graph during the measurement.</p> <p>If multiple layers are opened on the Data Display window, the scale change is applied to all layers. This means that the X-Y graph scale will be shared by all display layers. The auto scaling is performed as follows.</p> <ol style="list-style-type: none"><li>1. Graph scale is optimized for the data on the selected layer (top layer).</li><li>2. The graph scale is applied to the graph in all layers.</li></ol> <p>Note that the auto scaling is performed independently for the left and right Y axes individually.</p> <p>There is also a function for resetting the graph scale. Click View &gt; Cancel Scaling &gt; This Display or All Displays. This Display will reset the selected layer only. All Displays will reset for the all display layers.</p>
<b>Zoom In</b>	<p>This icon zooms the data graph in the cursor. If the cursor is not displayed, zooms the data graph in the last cursor position or the graph center. This enlarges the trace in the graph.</p>
<b>Zoom Out</b>	<p>This icon zooms the data graph out the cursor. If the cursor is not displayed, zooms the data graph out the last cursor position or the graph center. This reduces the trace in the graph.</p>
<b>Display Setup...</b>	<p>This icon opens “Display Setup (page 1-95)”.</p>
<b>Choose Active Y-axis</b>	<p>This icon selects the Y-axis that is effective for the marker control and the auto scale.</p>
<b>Marker ON/OFF</b>	<p>This icon enables or disables the marker.</p>
<b>Interpolation ON/OFF</b>	<p>This icon enables or disables the interpolation function of measurement data. You can read the interpolation data between two actual measurement points.</p>
<b>Marker Skip</b>	<p>This icon moves the marker to the next measurement curve that is added by the VAR2 variable.</p>
<b>Marker Maximum</b>	<p>This icon moves the marker to the maximum measurement point.</p>

<b>Marker Minimum</b>	This icon moves the marker to the minimum measurement point.
<b>Marker Search...</b>	This icon opens the Direct Marker/Cursor dialog box used to specify the coordinate values to move the marker.
<b>Line 1 State</b>	This icon enables or disables line 1 and its function: disabled, normal, gradient, tangent, regression, or fix.
<b>Line 2 State</b>	This icon enables or disables line 2 and its function: disabled, normal, gradient, tangent, regression, or fix.
<b>Cursor to Marker</b>	This icon moves the cursor to the marker position.
<b>Adjust Gradient</b>	This icon is available when a regression line is active. Clicking this icon enables the rotary knob and the mouse wheel to increase/decrease the gradient of the line. Clicking this icon again, pressing the rotary knob, or pressing the mouse wheel disables this function.
<b>Show Graph Legend</b>	This icon displays or hides the graph legend below the plot area.
<b>Show Line Information</b>	This icon displays or hides the line information (X-intercept, Y-intercept, gradient) in the plot area.

## Data Status

If measurement module detects any status while measurement, the status code is recorded with the measurement data. And the status code will be displayed in the Data Display window as shown in the following example.

### Example

- X-Y Graph Plot area

```
MARKER( 5.9000000 C 1.82520000 XV 3.33249E-3 )
```

This line will be displayed at the top of the plot area when marker is ON. The marker is used to read data of the marker point.

- List Display area

```
Index      V1              V2              I2
123        5.9 V          C 1.8252 V      XV 3.33249 mA
```

### Status Code

SMU status code:

- C: Compliance
- T: Other channel compliance
- X: Oscillation
- V: Overflow

MFCMU status code:

- U: Unbalance
- D: I/V amp saturation
- V: Overflow



## Setup Summary

This function is effective for the I/V Sweep and Multi Channel I/V Sweep classic tests. If the Setup Summary ON is set on the Graph Properties dialog box or the Data Display Properties dialog box, the channel setup summary information shown below is displayed at the bottom of the graph area.

The setup for one channel is displayed with one line as follows. Where, the items in { } are the setting values.

### VAR1 Channel

For the voltage output channel:

{V Name}: [{Start}, {Stop}] V / {No of Step}; {I Name}: {Compliance} A; {Pwr Comp} W

For the current output channel:

{I Name}: [{Start}, {Stop}] A / {No of Step}; {V Name}: {Compliance} V; {Pwr Comp} W

Example: VDS: [0, 10] V / 101; ID: 1 A; 1 W

### VAR2 Channel

For the voltage output channel:

{V Name}: [{Start}, {Stop}] V / {Step} V; {I Name}: {Compliance} A; {Pwr Comp} W

For the current output channel:

{I Name}: [{Start}, {Stop}] A / {Step} A; {V Name}: {Compliance} V; {Pwr Comp} W

Example: VGS: [0, 5] V / 1 V; ID: 1 mA; 1 mW

### Constant Channel

For the voltage output channel:

{V Name}: {Source} V; {I Name}: {Compliance} A

For the current output channel:

{I Name}: {Source} A; {V Name}: {Compliance} V

For the common channel:

{V Name}: COMMON

Example: VSUB: -1 V; ISUB: 1 mA

## Miscellaneous Operations

The following user interfaces provide several miscellaneous functions when you operate the Data Display window.

- “Mouse/Touch Panel”
- “Rotary Knob”
- “Softkeys”
- “Keyboard”

### Mouse/Touch Panel

While the Data Display window is focused, the mouse/touch panel operations shown in Table 1-11 (p. 1-106) are available.

### Rotary Knob

The following miscellaneous functions are available by using the rotary knob.

- Marker movement in the VAR1 direction

While the Data Display window is focused and the marker is displayed on the graph, the rotary knob can be used to move the marker. Rotating the knob fast increases the marker speed.

- Marker movement in the VAR2 direction

Clicking the knob moves the marker in the VAR2 direction.

- Slope adjustment of the gradient line

In the slope adjustment mode (Line > Adjust Gradient) for the gradient line, the rotary knob can be used to adjust the slope. Clicking the knob quits this mode.

- Pointer movement in the VAR1 direction

The rotary knob can be used to move the selected pointer. The pointer can not be moved in the VAR2 direction.

### Softkeys

The softkeys can be used for setting the following fields on the Display Setup (p. 1-95) window.

- Name (parameter name)
- Scale (Linear or Log)

## Keyboard

While the Data Display window is focused, the key operations shown in Table 1-6 are available.

**Table 1-6**

### Miscellaneous Key Operations on Data Display Window

Key	Description
F1	Displays the online help
F2	Shortcut to View > X-Y Graph
F3	Shortcut to View > List Display
F4	Shortcut to View > Parameters
F12	Changes the display mode, Append → Overwrite → Append
Ctrl+W	Shortcut to File > Close This Display
Ctrl+Shift+W	Shortcut to File > Close All Displays
Ctrl+S	Shortcut to File > Save Image As...
Ctrl+P	Shortcut to File > Print...
Ctrl+C	Shortcut to Edit > Copy

While the X-Y Graph (graph display area) is focused, the key operations shown in the following tables are available.

- Table 1-7, “Controlling X-Y Graph.”
- Table 1-8, “Controlling Marker.”
- Table 1-9, “Controlling Cursor.”
- Table 1-10, “Controlling Line.”

**Table 1-7 Controlling X-Y Graph**

Key	Description
S	Shortcut to View > Auto Scale
Z	Shortcut to View > Zoom In
Shift+Z	Shortcut to View > Zoom Out
Ctrl+Z	Shortcut to View > Cancel Scaling
Y	Changes the active data, Y1 → Y2 → ... → Y8 → Y1
R	Activates the right Y axis and changes the data set to the axis, Y2 → Y3 → ... → Y8 → Y2
W	Shortcut to View > Graph Legend ON/OFF
K	Shortcut to View > Line Information ON/OFF

**Table 1-8 Controlling Marker**

Key	Description
M	Shortcut to Marker > Marker ON/OFF
I	Shortcut to Marker > Interpolation ON/OFF
F	Moves the marker +1 <sup>a</sup> in the VAR1 direction
Shift+F	Moves the marker +10 <sup>a</sup> in the VAR1 direction
B	Moves the marker -1 <sup>a</sup> in the VAR1 direction
Shift+B	Moves the marker -10 <sup>a</sup> in the VAR1 direction
P	Moves the marker +1 <sup>a</sup> in the VAR2 direction
Shift+P	Moves the marker -1 <sup>a</sup> in the VAR2 direction
J	Shortcut to Marker > Go To...
X	Shortcut to Marker > Go To Maximum
N	Shortcut to Marker > Go To Minimum

a. Multiply 1/10 if the interpolation is ON.

**Table 1-9**

**Controlling Cursor**

<b>Key</b>	<b>Description</b>
C	Shortcut to Cursor > Cursor ON/OFF
T	Shortcut to Cursor > Go To Marker
↑	Moves the cursor +1/1000 in the Y axis direction
Shift+ ↑	Moves the cursor +1/100 in the Y axis direction
→	Moves the cursor +1/1000 in the X axis direction
Shift+ →	Moves the cursor +1/100 in the X axis direction
↓	Moves the cursor -1/1000 in the Y axis direction
Shift+ ↓	Moves the cursor -1/100 in the Y axis direction
←	Moves the cursor -1/1000 in the X axis direction
Shift+ ←	Moves the cursor -1/100 in the X axis direction

**Table 1-10**

**Controlling Line**

<b>Key</b>	<b>Description</b>
1 <sup>a</sup>	Shortcut to Line > Line 1 ON/OFF
2 <sup>a</sup>	Shortcut to Line > Line 2 ON/OFF
V	Changes the active line, Line 1 → Line 2 → Line 1 (available if both line 1 and line 2 are ON)
L	Changes the line mode of the active line, Normal → Gradient → Tangent → Regression → Normal
G	Shortcut to Line > Adjust Gradient

a. 1 and 2 on the numeric pad cannot be used.

**Table 1-11 Mouse/Touch Panel Operations on Data Display Window**

Location	Operation	Description
X-Y Graph	clicking on the left Y axis	Activates the Y1 data.
	clicking on the right Y axis	Activates the data of the right Y axis, or changes the active data if it is already active.
	clicking on a plot	Moves the marker to the position of the mouse pointer if the marker is displayed on the graph.
	clicking on a cursor	Specifies the active cursor if multiple cursors are displayed.
	dragging a cursor	Moves the cursor.
	clicking on a line	Specifies the active line if multiple lines are displayed.
	clicking on a text	Selects the text. Clicking with pressing the Shift key on the keyboard selects multiple texts.
	dragging a text	Moves the text.
	clicking on a pointer	Selects the pointer. Clicking with pressing the Shift key on the keyboard selects multiple pointers.
	clicking on a description of pointer	Selects the description of pointer. Clicking with pressing the Shift key on the key board selects multiple descriptions of pointers.
	dragging a description of pointer	Moves the description of pointer.
	rotating the mouse wheel	Moves the marker if the marker is displayed on the graph, moves the pointer if the pointer is selected, or adjusts the slope of the gradient line if it is in the slope adjustment mode (Line > Adjust Gradient).

Location	Operation	Description
X-Y Graph	clicking the mouse wheel	Same as clicking the rotary knob. This is effective if the mouse properties assign the center button functions to the wheel button.
List Display	dragging a column heading	Moves the column.
	dragging boundary of the column	Changes the width of the column.
	clicking on a row in the table	Moves the marker to the corresponding position.
	moving the scroll bar	Changes the data displayed in the List Display area. Scroll is effective for both portrait and landscape.
	rotating the mouse wheel	Changes the data displayed in the List Display area.
Others	clicking on a tab <sup>a</sup> under the tool bar	Specifies the display layer for the data display and analysis.
	double clicking on a tab <sup>a</sup> under the tool bar	Changes the status of the pin marked to the display layer. Standing pin must be marked to the display layer to be kept. Laid pin is marked to the display layer under the automatic delete.
	dragging boundary of the display area	Adjusts the size of the display area (X-Y Graph, List Display, Parameters) without changing the window size.
	dragging the frame of the Data Display window	Changes the window size with adjusting the size of the display area (X-Y Graph, List Display, Parameters).

- a. Tab is displayed if the display mode is Append and the test result record contains the append measurement data.

## Preview

The Preview window and the Preview Settings dialog box are opened by selecting View > Open Preview Window in the Data Display window. The Preview Settings dialog box is used to set the graph title, X axis title, Y axis title, and line style. And the Preview window displays a new plot, and is used to output it to the specified printer.

The Data Display window cannot do the above settings and display the new plot.

- “Preview Window GUI”
- “Preview Settings”

## Preview Window GUI

The Preview window provides the following GUI.

<b>Setup Name</b>	This field just displays the name of the test setup used to get this test record.
<b>X-Y Graph Plot</b>	This area displays the measurement result graph, traces, marker, cursor, and lines.
<b>Parameters</b>	This area lists the user-defined output parameters if the output parameters are set.
<b>List Display</b>	This area displays the measurement result data list.
<b>File menu</b>	Provides the following functions. See also Figure 1-7 and Figure 1-8.
<b>Page Setup...</b>	Opens a dialog box used to set the print page.
<b>Print Preview</b>	Opens a dialog box used to specify the data graphs to make previews. After selecting the graphs, previews appear on the Print preview window.
<b>Print...</b>	Opens a dialog box used to specify the print target. After selecting the target, the Print dialog box is opened. The dialog box is used to start print.
<b>Close Preview Window</b>	Closes the Preview window.



**Edit menu**

Provides the following functions. See also Figure 1-9 and Figure 1-10.

- Copy**
  - While the X-Y Graph Plot area is active, this function copies the graph image as BMP and EMF into the clipboard.
  - While the line cursor in the List Display area is active, this function copies the data list as CSV.
  - While the Parameters area is active, this function copies the parameter data as CSV.
- Copy Graph**
  - Copies the graph image as BMP and EMF into the clipboard.
- Copy List**
  - Copies the list data as CSV.
- Copy Parameters**
  - Copies the parameter data as CSV.

**View menu**

Provides the following functions.

- X-Y Graph**
  - Displays or hides the X-Y Graph Plot area.
- List Displays**
  - Displays or hides the List Display area.
- Parameters**
  - Displays or hides the Parameters area.
- Preview Settings...**
  - Opens “Preview Settings (page 1-110)”.

**Help menu**

- Agilent EasyEXPERT Help**
  - Opens the EasyEXPERT help window.

## Preview Settings

The Preview Settings dialog box provides the following GUI.

<b>General</b>	Defines the graph title in the Graph Title field. The title will appear at the bottom of the plot area.
<b>X/Y Axis Title</b>	Defines the X axis title in the X field and the Y axis title in the Y field, Y1 to Y8. The title will appear near the axis.
<b>Line Style</b>	Selects the line style for the line 1, line 2, and trace Y1 to Y8.
<b>Color and Thickness...</b>	Opens the Color and Thickness dialog box that provides the same GUI as “Graph Properties (page 1-96)”.
<b>OK</b>	Applies the setup to the Preview window, and closes this dialog box.
<b>Cancel</b>	Cancels the setup, and closes this dialog box.
<b>Apply</b>	Applies the setup to the Preview window.
<b>Reset</b>	Clears the setup and returns to the default setting.

## Text File Export settings

This dialog box is opened by clicking Results > Transport Data > Text File Export Settings... or the Settings button on the Test Results Data Folder Export dialog box or the Test Results Data Auto Export dialog box. This dialog box is used to define the style of the test record exported by the Export As Text File function.

- Reset** Applies the default value to the all setup items.
- OK** Defines and memorizes the present setting, and closes this dialog box.
- Cancel** Cancels and clears the setting changes, and closes this dialog box.
- Miscellaneous, Save As** Specifies the delimiter (data separator) and file extension. CSV (comma separated, .csv, default), Text (tab separated, .txt), or Formatted Text (space separated, .prn).
- Miscellaneous, Quotation** Specifies the quotation for the string values. Single (single quotes), Double (double quotes), or None (no quote, default). Even if None is specified, the string which contains delimiter and/or new line character will be double quoted.
- Identification section** Record items which are used to identify and characterize the test record. They are a sub set of the meta data, test type, and test name. See Table 1-13 for the definition of the record items. Check the box for the items you want to record, and uncheck the box for the items you do not want to record. In the default, all items are checked.
- The pull-down menu specifies the action for recording the items in the Identification section. See Table 1-12 to specify the action.

Table 1-12

**Actions for Identification and Setup Data Items**

Action	Description
Save with measurement data	The checked items are recorded in the test record with the measurement data. Default.
Do not save	No item is recorded even if any box is checked.
Save as separate file	The checked items are recorded in a separate file. The name will be the test record's file name plus -s before the extension.

Table 1-13

Identification Section Record Items

Item	Description
Setup title	Test setup name
Classic test name	Classic test name. I/V Sweep, Multi Channel I/V Sweep, I/V-t Sampling, C-V Sweep, SPGU Control, Switching Matrix Control, or Direct Control. This record item is exclusive against the “Application test name” record item.
Application test name	Application test name. This record item is exclusive against the “Classic test name” record item.
Test date	Test execution date corresponding to the date part of the RecordTime meta data.
Test time	Test execution time corresponding to the time part of the RecordTime meta data.
Device ID	Device ID corresponding to the TestTarget meta data.
Count	Test count corresponding to the IterationIndex meta data.
Flags	Flags set to the test result data, corresponding to the Flag meta data.
Remarks	Remarks set to the test result data, corresponding to the Remarks meta data.

**Setup data section** Record items which describes the test conditions. They are the test parameters, DUT parameters, and analysis setup. See Table 1-16 for the definition of the record items. Check the box for the items you want to record, and uncheck the box for the items you do not want to record.

Tag information for the setup data is string value; TestParameter, DutParameter, or AnalysisSetup, which means the type of data put in the same row in the exported test record. If *Tag information* box is unchecked (default), the tag information is not recorded. Then, the setup parameter names, such as Channel.Unit, are recorded in the first column in the test record and the parameter values are recorded in the following columns. If the *Tag information* box is checked, the tag information is recorded in the first column and the parameter names and values are shifted to the next columns.

The pull-down menu specifies the action for recording the items in the Setup data section. See Table 1-12 to specify the action.

**Measurement data section** Record items of the measurement data. It may contain the measurement parameter names, data units, and data size. See Table 1-17 for the definition of the record items. Check the box for the items you want to record, and uncheck the box for the items you do not want to record.

Tag information for the measurement data is string value; Dimension1, Dimension2, Data Name, Data Unit, or Data Value, which means the type of data put in the same row (in case of By Row orientation) or column (in case of By Column orientation) in the exported test record. The tag information will be recorded in the test record if the *Tag information* box is checked.

See Table 1-14 for the By Row orientation. All data relative to a measurement parameter will be put in the same column. The first column (ex. column A) contains the tag information. For the By Column orientation, see Table 1-15. All data relative to a measurement parameter will be put in the same row. And the first row (ex. row 1) contains the tag information. If the *Tag information* box is unchecked (default), the tag information is not recorded and the measurement data is shifted.

**Table 1-14 By Row Orientation Export Example**

	A	B	C	D	E	F
1	Dimension1	101	101	101	1	
2	Dimension2	1	1	1	1	
3	Data Name	Vg	Id	Gm	Vth	
4	Data Unit	V	A	S	V	
5	Data Value	0	4.55E-09	1.24E-10	1.23	
6	Data Value	0.02	8.59E-09	5.25E-10		

**Table 1-15 By Column Orientation Export Example**

	A	B	C	D	E	F
1	Dimension1	Dimension2	Data Name	Data Unit	Data Value	Data Value
2	101	1	Vg	V	0	0.02
3	101	1	Id	A	4.55E-09	8.59E-09
4	101	1	Gm	S	1.24E-10	5.25E-10
5	1	1	Vth	V	1.23	
6						

**Table 1-16 Setup Data Section Record Items (ON: checked, OFF: unchecked)**

Item	Description	Default
Test parameters	Test parameters defined by the classic test or the application test.	ON
DUT parameters	DUT parameters defined by the application test.	ON
Analysis setup	Data Display setup status at and after the measurement, such as Graph configuration, Graph format, Graph text, Marker configuration, Cursor configuration, Line configurations, Pointer configuration, List configuration, and Scalar list configuration.	OFF

**Table 1-17 Measurement Data Section Record Items (ON: checked, OFF: unchecked)**

Item	Description	Default
Data names	Measurement parameter name.	ON
Data units	Unit of measurement data.	OFF
Size	Dimension1 (VAR1 size) and Dimension2 (VAR2 size) of the measurement data.	OFF
Data values	Record items can be specified by the following segments.	
	<b>Segment</b>	<b>Default</b>
	Data assigned to X-Y Graph	ON
	Data assigned to List Display	ON
	Data assigned to Parameter Display	ON
	User function defined but not displayed	OFF
	Analysis function defined but not displayed	OFF
	Any data referred to by the user function but not directly displayed	OFF
Any data referred to by the analysis function but not directly displayed	OFF	



## Classic Test Definition

This chapter provides the reference information of the classic test setup GUI of Agilent EasyEXPERT software.

- “I/V Sweep”
- “Multi Channel I/V Sweep”
- “I/V List Sweep”
- “I/V-t Sampling”
- “C-V Sweep”
- “Direct Control”
- “Function Setup”
- “Auto Analysis Setup”
- “Display Setup”
- “SMU Range Setup Window”
- “ADC and Integration Time Setup Window”
- “Pulse Setup Window”
- “Advanced Setup Window”
- “Range Setup Window”
- “Advanced Setup Window for C-V Sweep”
- “Switching Matrix Control”
- “SPGU Control”
- “SPGU Pulse Setup Window”
- “Load Z Setup Window”
- “Pulse Switch Setup Window”
- “SPGU ALWG Setup Window”
- “Define ALWG Waveform Window”



To perform measurement, see “Main Screen” on page 1-11.

To display measurement data and perform data analysis, see “Data Display” on page 1-84.

To perform self-test or diagnosis, see “Configuration” on page 1-64.

To perform self-calibration, see “Calibration” on page 1-58.

---

**NOTE**

**If You Use GNDU (Ground Unit)**

Keep to the following rule to use the ground unit.

1. Set Mode to COMMON.
2. Set Function to CONST.
3. Do not use the I Name assigned to the ground unit.

Missing any one will cause an execution error.

---

## I/V Sweep

On the EasyEXPERT main screen, click the Classic Test tab and select the I/V Sweep icon. The main screen displays the I/V sweep test setup screen. For the measurement mode, see “I/V Sweep Measurement” on page 4-3.

<b>Setup Name</b>	Used to set the name of test setup. The name must be the identification for the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.  The name is recorded in the test result data and is used to differentiate the data in the test record list area.
<b>Channel Setup</b>	This tab displays the Channel Setup screen. See “Channel Setup” on page 2-5.
<b>Measurement Setup</b>	This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-6.
<b>Function Setup</b>	This tab displays the Function Setup screen. See “Function Setup” on page 2-42.
<b>Auto Analysis Setup</b>	This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.
<b>Display Setup</b>	This tab displays the Display Setup screen. See “Display Setup” on page 2-46.

## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, measurement data, time stamp data, and index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data and the measurement data.

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	Output mode. For SMU, V (voltage), I (current), VPULSE (voltage pulse), IPULSE (current pulse), or COMMON (ground). For SPGU, VPULSE (voltage pulse) or ALWG (arbitrary linear waveform voltage).
<b>Function</b>	Output function, VAR1 (primary sweep), VAR2 (secondary sweep), VAR1' (synchronous sweep), or CONST (constant)
<b>Add SMU</b>	Adds a row for the SMU channel setup.
<b>Add SPGU</b>	Adds a row for the SPGU channel setup.
<b>Delete</b>	Deletes the channel setup. To select the channel setup, use the left radio button.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. The output stops in the opposite order.

### Miscellaneous Variables

This area defines the variables for the time stamp data and the index data.

<b>Time Stamp Name</b>	Variable name of the time stamp. Time stamp data gives the time when step measurement is started.
<b>Index Name</b>	Variable name of the data index. Integer above 0 is returned.

## Measurement Setup

This setup screen is used to set the source channel outputs, the measurement ranging mode, the sweep abort condition, the measurement channel A/D converter, the series resistor, the filter, and the source output/measurement wait time.

### VAR1

This area defines the primary sweep source output.

<b>Unit</b>	SMU used for the VAR1 output
<b>Name</b>	Variable name for the VAR1 output data
<b>Direction</b>	Sweep direction, single (start to stop) or double (start to stop to start)
<b>Linear/Log</b>	Sweep output, LINEAR, LOG10, LOG25, or LOG50. where the number following LOG means the number of sweep points in one decade.
<b>Start</b>	Sweep start value
<b>Stop</b>	Sweep stop value
<b>Step</b>	Incremental step value of the sweep output
<b>No of Step</b>	Number of steps. The value is automatically calculated by using the start, stop, and step values.
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

**VAR2**

This area defines the secondary sweep source output.

<b>Unit</b>	SMU used for the VAR2 output
<b>Name</b>	Variable name for the VAR2 output data
<b>Start</b>	Sweep start value
<b>Stop</b>	Sweep stop value. The value is automatically calculated by using the start, step, and number of step values.
<b>Step</b>	Incremental step value of the sweep output
<b>No of Step</b>	Number of sweep steps
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

**VAR1'**

This area defines the synchronous sweep source output. The VAR1' output is given by the following formula.

$$VAR1' \text{ output} = \text{Offset} + \text{Ratio} \times VAR1 \text{ output}$$

<b>Unit</b>	SMU used for the VAR1' output
<b>Name</b>	Variable name for the VAR1' output data
<b>Offset</b>	Offset value
<b>Ratio</b>	Ratio value
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

**Timing**

This area defines the hold time and the delay time. See “To Specify Source Output Time” on page 4-67.

<b>Hold</b>	Hold time. This is the time from beginning of channel output to beginning of delay time. 0 to 655.35 s, 10 ms resolution.
<b>Delay</b>	Delay time. This is the time from end of hold time to beginning of measurement. The measurement channels start measurement as soon as the wait time elapses. 0 to 65.535 s, 0.1 ms resolution.

## Classic Test Definition

### I/V Sweep

<b>Constant</b>	This area defines the constant output source.
<b>Unit</b>	SMU used for the constant source
<b>V Name</b>	Variable name for the constant voltage output or measurement data
<b>I Name</b>	Variable name for the constant current output or measurement data
<b>Mode</b>	Output mode, V (voltage), I (current), VPULSE (voltage pulse), or IPULSE (current pulse)
<b>Source</b>	Output value
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>SMU Pulse</b>	This area defines the SMU pulse setup parameters. See “SMU Pulse” on page 4-62.
<b>Unit</b>	SMU used for the pulse output
<b>Period</b>	Pulse period. Entering AUTO sets the optimum value automatically.
<b>Width</b>	Pulse width
<b>Base</b>	Pulse base value
<b>Sweep status</b>	This field specifies the sweep abort condition. See “Sweep Abort Function” on page 4-37.
<b>Range</b>	This button opens a window used to set the measurement ranging operation. See “SMU Range Setup Window” on page 2-48.
<b>ADC/Integ</b>	This button opens a window used to set the A/D converter of the measurement channels. See “ADC and Integration Time Setup Window” on page 2-50.
<b>Advanced</b>	This button opens a window used to set the other functions of the measurement channel. See “Advanced Setup Window” on page 2-53.
<b>SPGU Pulse / ALWG Setup</b>	This button opens a window used to set the SPGU output. See “SPGU Pulse Setup Window” on page 2-63 or “SPGU ALWG Setup Window” on page 2-67.

---

## Multi Channel I/V Sweep

On the EasyEXPERT main screen, click the Classic Test tab and select the Multi Channel I/V Sweep icon. The main screen displays the Multi Channel I/V Sweep test setup screen. For the measurement mode, see “Multi Channel I/V Sweep Measurement” on page 4-12.

<b>Setup Name</b>	<p>Sets the name of the test setup. The name is used to identify the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.</p> <p>The name is recorded in the test result data and is used to differentiate the data in the test record list area.</p>
<b>Channel Setup</b>	<p>This tab displays the Channel Setup screen. See “Channel Setup” on page 2-10.</p>
<b>Measurement Setup</b>	<p>This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-11.</p>
<b>Function Setup</b>	<p>This tab displays the Function Setup screen. See “Function Setup” on page 2-42.</p>
<b>Auto Analysis Setup</b>	<p>This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.</p>
<b>Display Setup</b>	<p>This tab displays the Display Setup screen. See “Display Setup” on page 2-46.</p>

---

### NOTE

Setting of the Display Setup is applied to the X-Y Graph on the Data Display window. And the X-Y Graph can handle one X data for the X axis and the maximum eight Y data for the Y axis.

Define the X axis carefully to plot the Multi Channel I/V Sweep measurement result which may have the data measured by the multiple VAR1 channels. Because the result may also contain the multiple X data which can be the selections of the X axis.

The Display Setup must be set properly to match the X axis data and the X data of the plot you look at.

## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, measurement data, time stamp data, and index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data and the measurement data.

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	Output mode. For SMU, V (voltage), I (current), VPULSE (voltage pulse), IPULSE (current pulse), or COMMON (ground). For SPGU, VPULSE (voltage pulse) or ALWG (arbitrary linear waveform voltage).
<b>Function</b>	Output function, VAR1 (primary sweep), VAR2 (secondary sweep), or CONST (constant)
<b>Add SMU</b>	Adds a row for the SMU channel setup.
<b>Add SPGU</b>	Adds a row for the SPGU channel setup.
<b>Delete</b>	Deletes selected channel setup. To select a channel, use the radio button on the left.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. The output stops in the opposite order.

### Miscellaneous Variables

This area defines the variables for the time stamp data and the index data.

<b>Time Stamp Name</b>	Variable name of the time stamp. Time stamp data indicates the time at which step measurement was started.
<b>Index Name</b>	Variable name of the data index. Integer above 0 is returned.



## Measurement Setup

This setup screen is used to set the source channel outputs, the measurement ranging mode, the sweep abort condition, the measurement channel A/D converter, the series resistor, the filter, and the source output/measurement wait time.

### VAR1

This area defines the primary sweep source output.

<b>Direction</b>	Sweep direction, single (start to stop) or double (start to stop to start)
<b>Linear/Log</b>	Sweep output, LINEAR, LOG10, LOG25, or LOG50 where the number following LOG indicates the number of sweep points in one decade.
<b>No of Step</b>	Number of sweep steps
<b>Unit</b>	SMU used for the VAR1 output
<b>Name</b>	Variable name for the VAR1 output data
<b>Start</b>	Sweep start value
<b>Stop</b>	Sweep stop value
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

### Constants

This area defines the constant output source.

<b>Unit</b>	SMU used for the constant source
<b>V Name</b>	Variable name for the constant voltage output or measurement data
<b>I Name</b>	Variable name for the constant current output or measurement data
<b>Mode</b>	Output mode, V (voltage) or I (current)
<b>Source</b>	Output value
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.

### VAR2

This area defines the secondary sweep source output.

<b>Unit</b>	SMU used for the VAR2 output
-------------	------------------------------

## Classic Test Definition

### Multi Channel I/V Sweep

<b>Name</b>	Variable name for the VAR2 output data
<b>Start</b>	Sweep start value
<b>Stop</b>	Sweep stop value. The value is automatically calculated from the start, step, and number of step values.
<b>Step</b>	Incremental step value for the sweep output
<b>No of Step</b>	Number of sweep steps
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

**Timing** This area defines the hold time and the delay time. See “To Specify Source Output Time” on page 4-67.

**Hold** Hold time. This is the time between the beginning of channel output and the beginning of delay time. 0 to 655.35 s, 10 ms resolution.

**Delay** Delay time. This is the time between the end of the hold time and the beginning of measurement. The measurement channels start measurement as soon as the wait time elapses. 0 to 65.535 s, 0.1 ms resolution.

**Sweep status** This field specifies the sweep abort condition. See “Sweep Abort Function” on page 4-37.

**Range** This button opens a window for setting measurement ranging operation. See “SMU Range Setup Window” on page 2-48.

**ADC/Integ** This button opens a window for setting the A/D converter of the measurement channels. See “ADC and Integration Time Setup Window” on page 2-50.

**Pulse** This button opens a window for setting the SMU pulse output. See “Pulse Setup Window” on page 2-51.

**Advanced** This button opens a window for setting other functions of the measurement channel. See “Advanced Setup Window” on page 2-53.

**SPGU Pulse / ALWG Setup** This button opens a window used to set the SPGU output. See “SPGU Pulse Setup Window” on page 2-63 or “SPGU ALWG Setup Window” on page 2-67.

---

## I/V List Sweep

On the EasyEXPERT main screen, click the Classic Test tab and select the I/V Sweep icon. The main screen displays the I/V List sweep test setup screen.

I/V List Sweep measurement is similar to I/V Sweep measurement described in “I/V Sweep Measurement” on page 4-3.

Changes from the I/V Sweep are listed below.

- VAR1/VAR2 step output values are set by using a vector data. Vector data from a spreadsheet software, either tab separated or comma separated can be copied to the Define vector data dialog box via the Windows clipboard. See “Define vector data” on page 2-18.
- All SMU installed in the B1500 can be set to VAR1.
- VAR1’ is not supported.

<b>Setup Name</b>	Sets the name of the test setup. The name is used to identify the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.  The name is recorded in the test result data and is used to differentiate the data in the test record list area.
<b>Channel Setup</b>	This tab displays the Channel Setup screen. See “Channel Setup” on page 2-5.
<b>Measurement Setup</b>	This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-6.
<b>Function Setup</b>	This tab displays the Function Setup screen. See “Function Setup” on page 2-42.
<b>Auto Analysis Setup</b>	This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.
<b>Display Setup</b>	This tab displays the Display Setup screen. See “Display Setup” on page 2-46.

## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, the measurement data, the time stamp data, and the index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data and the measurement data.

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	Output mode. For SMU, V (voltage), I (current), VPULSE (voltage pulse), IPULSE (current pulse), or COMMON (ground). For SPGU, VPULSE (voltage pulse) or ALWG (arbitrary linear waveform voltage).
<b>Function</b>	Output function, VAR1 (primary sweep), VAR2 (secondary sweep), or CONST (constant).  If a SMU is set to Mode=VPULSE or IPULSE: <ul style="list-style-type: none"><li>• Number of VAR1 channels must be one or two.</li><li>• If a VAR1 channel is set to Mode=VPULSE, the other VAR1 channel must be Mode=V.</li><li>• If a VAR1 channel is set to Mode=IPULSE, the other VAR1 channel must be Mode=I.</li></ul>
<b>Add SMU</b>	Adds a row for the SMU channel setup.
<b>Add SPGU</b>	Adds a row for the SPGU channel setup.
<b>Delete</b>	Deletes the selected channel setup. To select a channel setup, use the radio button on the left.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. The output stops in the opposite order.

## Miscellaneous Variables

This area defines the variables for the time stamp data and the index data.

**Time Stamp Name** Variable name of the time stamp. Time stamp data indicates the time at which step measurement was started.

**Index Name** Variable name of the data index. Integer above 0 is returned.

## Measurement Setup

This setup screen is used to set the source channel outputs, the measurement ranging mode, the sweep abort condition, the measurement channel A/D converter, the series resistor, the filter, and the source output/measurement wait time.

### VAR1

This area defines the primary sweep source output.

**Unit** SMU used for the VAR1 output

**Source** Click the grid button to open the Define vector data dialog box. See “Define vector data” on page 2-18.

**Compliance** Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.

**Pwr Comp** Power compliance value (in W), or OFF

---

### NOTE

#### To define VAR1/VAR2 step outputs

To define these parameters, create a vector data by using the Define vector data dialog box which is opened by clicking the grid button in the vector data entry field. See Figure 2-1.

Available array size for the vector data is as follows (column × row).

- VAR1 step output value:  $N_1 = 1 \times 1001$  maximum (initial setting:  $1 \times 1$ )
- VAR2 step output value:  $N_2 = 1 \times 1001$  maximum (initial setting:  $1 \times 1$ )
- Delay time:  $N_3 = 1 \times 1001$  maximum (initial setting:  $1 \times 1$ )

Then, the following formula must be satisfied.

$$1 \leq N_1 \times N_2 \leq 128128$$

---

## Classic Test Definition

### I/V List Sweep

#### VAR2

This area defines the secondary sweep source output.

<b>Unit</b>	SMU used for the VAR2 output
<b>Name</b>	Variable name for the VAR2 output data
<b>Source</b>	Click the grid button to open the Define vector data dialog box. See “Define vector data” on page 2-18.
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Pwr Comp</b>	Power compliance value (in W), or OFF

#### Timing

This area defines the hold time and the delay time. See “To Specify Source Output Time” on page 4-67.

<b>Hold</b>	Hold time. This is the time from the beginning of channel output to the beginning of delay time. 0 to 655.35 s, 10 ms resolution.
<b>Delay</b>	Delay time. This is the time between end of step changing and the beginning of measurement. For the first step, this is the time between the end of hold time and the beginning of measurement. The measurement channels start measurement as soon as the wait time elapses. 0 to 65.535 s, 0.1 ms resolution. Click the grid button to open the Define vector data dialog box. See “Define vector data” on page 2-18.

#### SMU Pulse

This area defines the SMU pulse setup parameters. See “SMU Pulse” on page 4-62.

<b>Unit</b>	SMU used for the pulse output
<b>Period</b>	Pulse period. Cannot set this value. SMU applies the pulse after the delay time immediately.
<b>Width</b>	Pulse width
<b>Base</b>	Pulse base value

<b>Constant</b>	This area defines the constant output source.
<b>Unit</b>	SMU used for the constant source
<b>V Name</b>	Variable name for the constant voltage output or measurement data
<b>I Name</b>	Variable name for the constant current output or measurement data
<b>Mode</b>	Output mode, V (voltage), I (current), VPULSE (voltage pulse), or IPULSE (current pulse)
<b>Source</b>	Output value
<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Sweep status</b>	This field specifies the sweep abort condition. See “Sweep Abort Function” on page 4-37.
<b>Range</b>	This button opens a window for setting the measurement ranging operation. See “SMU Range Setup Window” on page 2-48.
<b>ADC/Integ</b>	This button opens a window for setting the A/D converter of the measurement channels. See “ADC and Integration Time Setup Window” on page 2-50.
<b>Advanced</b>	This button opens a window for setting other functions of the measurement channel. See “Advanced Setup Window” on page 2-53.
<b>SPGU Pulse / ALWG Setup</b>	This button opens a window used to set the SPGU output. See “SPGU Pulse Setup Window” on page 2-63 or “SPGU ALWG Setup Window” on page 2-67.

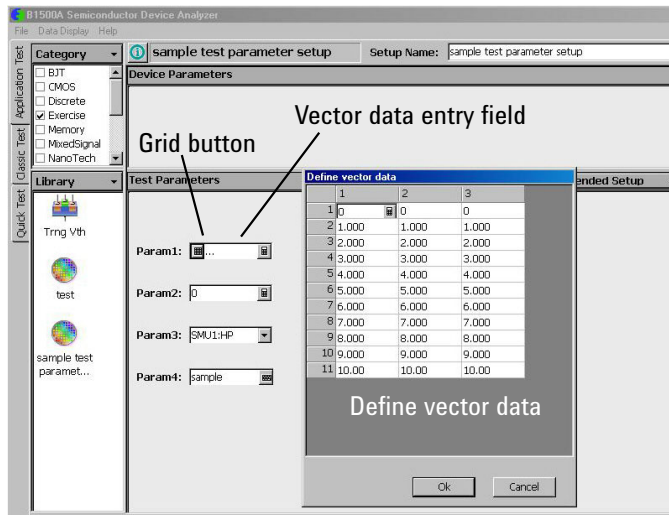
## Define vector data

This dialog box is opened by clicking the grid button in the vector data entry field. See Figure 2-1. The vector data entry field can be displayed on the followings.

- Application test setup screen which contains the entry field for a vector type test parameter, on the EasyEXPERT main screen or the Test Definition window
- Test Specification setup screen which contains the definition of a vector type test parameter, on the Test Definition window
- Define symbols dialog box for a vector type test parameter

The Define vector data dialog box displays the array defined by using the Define dimensions dialog box opened from the Test Specification setup screen. Figure 2-1 shows the array created by setting Dim1Min=1, Dim1Max=11, Dim2Min=1, and Dim2Max=3, where Dim1 is the vertical direction (row) and Dim2 is the horizontal direction (columns). The array index always starts from 1.

Figure 2-1 Vector Data Entry Field



### NOTE

Rotary knob and mouse wheel can be used to change the numeric value in the highlighted cell.

To delete column/row, highlight the line and press the Delete key.



The Define vector data dialog box provides the following GUI.

<b>OK button</b>	Saves the vector data displayed on the dialog box and closes the dialog box.
<b>Cancel button</b>	Displays a confirmation message and closes the dialog box. Vector data is not saved.
<b>File menu</b>	Provides the following function. <b>Exit</b> Displays a confirmation message and closes the dialog box. Vector data is not saved.
<b>Edit menu</b>	Provides the following four functions. <b>Copy</b> Copies the highlighted item to the clipboard. <b>Paste</b> Pastes the data in the clipboard to the specified location. <b>Delete</b> Deletes the highlighted item. <b>Select All</b> Selects and highlights all cells in the array.
<b>Mouse menu 1</b>	The following menus are displayed when the right mouse button is clicked on a cell. <b>Cut</b> Moves the highlighted characters to the clipboard. <b>Copy</b> Copies the highlighted characters to the clipboard. <b>Paste</b> Pastes the data in the clipboard to the specified location.
<b>Mouse menu 2</b>	The following menus are displayed when the right mouse button is clicked on the array display area. <b>Copy</b> Copies the highlighted item to the clipboard. <b>Paste</b> Pastes the data in the clipboard to the specified location. <b>Insert</b> Inserts a row before the selected row. Available when the row header provides the * button and a row header is selected. <b>Delete</b> Deletes the highlighted item. <b>Select All</b> Selects and highlights all cells in the array.

## Classic Test Definition

### I/V List Sweep

#### **To import vector data**

The Define vector data dialog box can import comma separated data (each data must be separated by a comma, and have a CR/LF or LF at the end of each line) and tab separated data (each data must be separated by a tab, and have a CR/LF or LF at the end of each line). The procedure for importing vector data is as follows.

1. Open a vector data with Notepad.
2. Copy the data in Notepad.
3. On the Define vector data dialog box, click a cell to be used as the top left corner of the imported vector data.
4. Right click on the corner box between the column header and the row header, and select Paste.

Thus, the Define vector data dialog box can read vector data created on a spreadsheet software.

#### **To export vector data**

The Define vector data dialog box can export vector data in tab separated format. The procedure for exporting vector data is as follows.

1. Highlight a vector data on the Define vector data dialog box.
2. Select Copy from the Edit menu or the mouse menu 2.
3. Create a new file in Notepad, and paste the vector data.
4. Save the data as a file.

The data can then be read by opening the file with a spreadsheet software.

---

## I/V-t Sampling

On the EasyEXPERT main screen, click the Classic Test tab and select the I/V-t Sampling icon. The main screen displays the I/V-t sampling test setup screen. For details of the measurement mode, see “I/V-t Sampling Measurement” on page 4-13.

### Setup Name

Used to set the name of test setup. The name must be the identification for the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is recorded in the test result data and is used to differentiate the data in the test record list area.

### Channel Setup

This tab displays the Channel Setup screen. See “Channel Setup” on page 2-22.

### Measurement Setup

This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-23.

### Function Setup

This tab displays the Function Setup screen. See “Function Setup” on page 2-42.

### Auto Analysis Setup

This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.

### Display Setup

This tab displays the Display Setup screen. See “Display Setup” on page 2-46.

---

### NOTE

If you set Interval < 0.002 s

Sampling mode must be linear. This setting is not permitted for the log sampling. Also SPGU is not available.

The following conditions are automatically set to the all measurement channels. And the all channels start measurement simultaneously. After the measurement, they are returned to the previous conditions automatically.

- High speed A/D converter (ADC)
- Minimum measurement range that covers compliance value, fixed ranging

If the measurement time is expected to be longer than *interval*, the measurement channels automatically adjust the number of averaging samples (ADC settings) to keep the sampling interval. After the measurement, it is returned to the previous setting automatically.

## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, measurement data, time stamp data, and index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data and the measurement data.

---

**NOTE** UHCU and HVMCU cannot be used.

---

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	Output mode. For SMU, V (voltage), I (current), or COMMON (ground). For SPGU, VPULSE (voltage pulse) or ALWG (arbitrary linear waveform voltage).
<b>Add SMU</b>	Adds a row for the SMU channel setup.
<b>Add SPGU</b>	Adds a row for the SPGU channel setup.
<b>Delete</b>	Deletes the channel setup. To select the channel setup, use the left radio button.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. The output stops in the opposite order. However, if the Output Sequence is SIMULTANEOUS, the channels start and stop output simultaneously.

### Miscellaneous Variables

This area defines the variables for the time stamp data and the index data.

<b>Time Stamp Name</b>	Variable name of the time stamp. Time stamp data gives the time when step measurement is started.
<b>Index Name</b>	Variable name of the data index. Integer above 0 is returned.

## Measurement Setup

This setup screen is used to set the sampling timing parameters, constant voltage/current outputs, and so on.

### Sampling Parameter

This area defines the sampling timing parameters.

<b>Linear/Log</b>	<p>Sampling mode. LINEAR for linear sampling. LOG10, LOG25, LOG50, LOG100, LOG250, or LOG500 for logarithmic sampling.</p> <p>For the logarithmic sampling, the number after LOG indicates the number of measurement data in a decade. For example, LOG10 mode is used to get 10 data/decade.</p>
<b>Interval</b>	<p>Interval of the sampling, in seconds. 0.002 to 65.535 s, 0.001 s resolution.</p> <p><i>Interval</i> &lt; 0.002 s in 0.00001 s resolution is also available for the linear sampling. It must satisfy the following formula. See NOTE on page 2-21.</p> $Interval \geq 0.0001 + 0.00002 \times (\text{number of measurement channels} - 1)$
<b>No of Samples</b>	<p>Number of samples. Integer. 1 to the following value.</p> <p>For linear: <math>100001 / (\text{number of measurement channels})</math></p> <p>For logarithmic: <math>1 + (\text{number of data for 11 decades})</math></p>
<b>Total Sampling Time</b>	<p>This field just displays the total sampling time that is the time from the measurement start time for the first point to the end of sampling measurement. It does not include the hold time.</p> $Total\ Sampling\ Time = Interval \times No\ of\ Samples$
<b>Output Sequence</b>	<p>Source output sequence. SIMULTANEOUS or SEQUENTIAL. See “Source Output Sequence and Time Origin” on page 4-18.</p>
<b>Hold Time</b>	<p>Time since the <i>Source</i> value output until the first sampling point, in seconds. 0 to 655.35 s, resolution 0.01 s.</p> <p>The following values are also available for <i>Interval</i> &lt; 0.002 s. <math> Hold\ Time </math> will be the time since the sampling start until the <i>Source</i> value output.</p> <p>-0.09 to -0.0001 s, resolution 0.0001 s.</p>

## Classic Test Definition

### I/V-t Sampling

**Base Hold Time** Hold time of the *Base* value output until the *Source* value output, in seconds. 0 to 655.35 s, resolution 0.01 s.

**Stop Condition** This area defines the sampling measurement stop condition. See “Stop Condition” on page 4-18 for this function.

**Enable/Disable** Enables or disables the stop condition.

**Enable Delay** Delay time. in second. This is the time from starting sampling measurement to enabling this function.

**Name** Name of measurement data or user function to monitor for stop condition. Val of Event.

Cannot use the user function that uses the following built-in functions.

at, avg, cond, dim1Size, dim2Size, integ, mavg, max, min, storeAt

**Threshold** Threshold value at which to stop sampling measurement. Th of Event.

**Event** Event for stop condition.

**Val > Th** True if Name parameter value is greater than Threshold value.

**Val < Th** True if Name parameter value is less than Threshold value.

**|Val| > |Th|** True if absolute Name parameter value is greater than absolute Threshold value.

**|Val| < |Th|** True if absolute Name parameter value is less than absolute Threshold value.

**No. of Events** Target value of the count the event occurs (true). When the count of true is this value, sampling is immediately stopped.

**Constants** This area defines the constant voltage/current source.

**Unit** SMU used for the constant source

**V Name** Variable name for the constant voltage output or measurement data

**I Name** Variable name for the constant current output or measurement data

<b>Mode</b>	Output mode, V (voltage), or I (current)
<b>Base, Source</b>	Base or Source value, in V or A.  <i>Base</i> value is available for all output channels for the SIMULTANEOUS mode and the last source channel for the SEQUENTIAL mode.
<b>Compliance</b>	Compliance value, in A or V.

**Range** This button opens a window used to set the measurement ranging operation. See “SMU Range Setup Window” on page 2-48.

**ADC/Integ** This button opens a window used to set the A/D converter of the measurement channels. See “ADC and Integration Time Setup Window” on page 2-50.

**Advanced** This button opens a window used to set the other functions of the measurement channel. See “Advanced Setup Window” on page 2-53.

**SPGU Pulse / ALWG Setup** This button opens a window used to set the SPGU output. See “SPGU Pulse Setup Window” on page 2-63 or “SPGU ALWG Setup Window” on page 2-67.  
SPGU cannot be used for the sampling measurement of *Interval* < 0.002 s.

## C-V Sweep

On the EasyEXPERT main screen, click the Classic Test tab and select the C-V Sweep icon. The main screen displays the C-V sweep test setup screen.

For details of the measurement mode, see “C-V Sweep Measurement” on page 4-19.

### Setup Name

Used to set the name of test setup. The name must be the identification for the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is recorded in the test result data and is used to differentiate the data in the test record list area.

### Channel Setup

This tab displays the Channel Setup screen. See “Channel Setup” on page 2-27.

### Measurement Setup

This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-28.

### Function Setup

This tab displays the Function Setup screen. See “Function Setup” on page 2-42.

### Auto Analysis Setup

This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.

### Display Setup

This tab displays the Display Setup screen. See “Display Setup” on page 2-46.

---

### NOTE

#### Error Correction

Error correction must be performed before starting measurements. See “CMU Calibration” on page 1-60 that explains the error correction GUI. There are four techniques. For the easy way, perform the phase compensation and the open correction.

---



## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, time stamp data, and index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data.

**NOTE** UHCU and HVMCU cannot be used.

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of voltage output or measurement data
<b>I Name</b>	Variable name of current output or measurement data
<b>Mode</b>	SMU output mode, V (voltage), I (current), or COMMON (ground)
<b>Function</b>	Output function, VAR1 (primary sweep) or CONST (constant)
<b>Add</b>	Adds a row for the channel setup.
<b>Delete</b>	Deletes the channel setup. To select the channel setup, use the left radio button.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

If multiple output channels are defined, the channels start output sequentially. The order is top to bottom of the channels defined in this screen. However the last output channel is always the MFCMU. The output stops in the opposite order.

**Miscellaneous Variables** This area defines the variables for the time stamp data and the index data.

<b>Time Stamp Name</b>	Variable name of the time stamp. Time stamp data gives the time when step measurement is started.
<b>Index Name</b>	Variable name of the data index. Integer above 0 is returned.

## Measurement Setup

This setup screen is used to set the MFCMU measurement mode, source channel outputs, the sweep abort condition, and the measurement time.

### C-V (VAR1)

This area defines the MFCMU measurement mode and DC bias output.

<b>V Name</b>	Variable name of the DC bias output data
<b>Model</b>	MFCMU measurement mode. Primary and secondary measurement parameters. See Table 4-2 on page 4-21. Select one from the pull-down menu.
<b><i>para1</i> Name</b>	Variable name of the primary measurement parameter
<b><i>para2</i> Name</b>	Variable name of the secondary measurement parameter
<b>Direction</b>	Sweep direction, single (start to stop) or double (start to stop to start)
<b>Linear/Log</b>	Sweep output, LINEAR, LOG10, LOG25, or LOG50. where the number following LOG means the number of sweep points in one decade.
<b>Start</b>	Sweep start value
<b>Stop</b>	Sweep stop value
<b>Step</b>	Incremental step value of the sweep output
<b>Compliance</b>	Only for SMU. Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>No of Step</b>	Number of steps. The value is automatically calculated by using the start, stop, and step values.

### Signal Source

This area defines the measurement signal.

<b>Frequency List</b>	Frequency of the measurement signal
<b>F Name</b>	Variable name of the frequency data
<b>AC Level</b>	Oscillator level of the measurement signal, 10 mV to 250 mV, 1 mV resolution
<b>Add</b>	Adds a measurement frequency.
<b>Delete</b>	Deletes the selected frequency. To select the frequency, use the left radio button.

	<b>Up</b>	Moves the selected frequency upward.
	<b>Down</b>	Moves the selected frequency downward.
<b>Integration Time</b>		This area sets the A/D converter of MFCMU.
	<b>Mode</b>	A/D converter operation mode, AUTO or PLC
	<b>Factor</b>	Factor for the initial value. See “Mode, Factor” on page 4-20.
<b>Timing</b>		This area defines the hold time and the delay time. See “To Specify Source Output Time” on page 4-67.
	<b>Hold</b>	Hold time. This is the time from beginning of channel output to beginning of delay time. 0 to 655.35 s, 10 ms resolution.
	<b>Delay</b>	Hold time. This is the time from end of hold time to beginning of measurement. The MFCMU starts measurement as soon as the wait time elapses. 0 to 655.35 s, 0.1 ms resolution.
<b>Constant</b>		This area defines the constant voltage/current source.
	<b>Unit</b>	SMU used for the constant source
	<b>V Name</b>	Variable name for constant voltage output or measurement data
	<b>I Name</b>	Variable name for constant current output or measurement data
	<b>Mode</b>	Output mode, V (voltage), I (current), or COMMON (ground).
	<b>Source</b>	Output value
	<b>Compliance</b>	Compliance value. Enter the current limit value for the voltage source, or the voltage limit value for the current source.
<b>Sweep status</b>		This field specifies the sweep abort condition. See “Sweep Abort Function” on page 4-37.
<b>Range</b>		This button opens a window used to set the measurement ranging operation. See “Range Setup Window” on page 2-55.
<b>ADC/Integ</b>		This button opens a window used to set the A/D converter of the SMU measurement channels. See “ADC and Integration Time Setup Window” on page 2-50.
<b>Advanced</b>		This button opens a window used to set the other functions of the measurement channel. See “Advanced Setup Window for C-V Sweep” on page 2-57.

## Direct Control

On the EasyEXPERT main screen, click the Classic Test tab and select the Direct Control icon. The main screen displays the Direct Control test setup screen.

Direct Control test mode performs measurement by controlling measurement resources, SMU, CMU, and SPGU directly. The instrument can be controlled by using GPIB commands. For the GPIB commands, see *Agilent B1500 Programming Guide*.

Available GPIB commands are listed in the Command pull-down menu on the Measurement Setup screen. Other commands are not supported by the Direct Control test.

<b>Setup Name</b>	<p>Sets the name of the test setup. The name is used to identify the test setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.</p> <p>The name is recorded in the test result data and is used to differentiate the data in the test record list area.</p>
<b>Channel Setup</b>	<p>This tab displays the Channel Setup screen. See “Channel Setup” on page 2-31.</p>
<b>Measurement Setup</b>	<p>This tab displays the Measurement Setup screen. See “Measurement Setup” on page 2-33.</p>
<b>Function Setup</b>	<p>This tab displays the Function Setup screen. See “Function Setup” on page 2-42.</p> <p>If Agilent B2200A/B2201A/E5250A switching matrix is used, the CMU measurement data should be compensated. To perform the compensation, define a user function by using the “compenReZ” and “compenImZ” functions or the “compenReY” and “compenImY” functions.</p>
<b>Auto Analysis Setup</b>	<p>This tab displays the Auto Analysis Setup screen. See “Auto Analysis Setup” on page 2-43.</p>
<b>Display Setup</b>	<p>This tab displays the Display Setup screen. See “Display Setup” on page 2-46.</p>

## Channel Setup

This setup screen is used to define the channel setups and the variables for the channel output data, measurement data, time stamp data, and index data.

**Channel Definition** This area defines the channel setups and the variables for the channel output data and the measurement data.

<b>Unit</b>	Measurement resource used for the source output or measurement
<b>V Name</b>	Variable name of DC voltage output or measurement data
<b>I Name</b>	Variable name of DC current output or measurement data
<b>Add SMU</b>	Adds a row for the SMU channel setup.
<b>Add CMU</b>	Adds a row for the CMU channel setup.
<b>Add SPGU</b>	Adds a row for the SPGU channel setup.
<b>Delete</b>	Deletes the selected channel setup. To select a channel setup, use the radio button on the left.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

**Additional Data Variables** This area defines the variables for the measurement data other than DC voltage and DC current which can be defined in Channel Definition.

<b>Unit</b>	Unit (SMU or CMU) used for the measurement
<b>Type</b>	Type of measurement data. See Table 2-1.
<b>Name</b>	Variable name of the measurement data
<b>Add</b>	Adds a row for the measurement variable.
<b>Delete</b>	Deletes the selected measurement variable. To select a variable, use the radio button on the left.

**Miscellaneous Variables** This area defines the variables for the time stamp data and the index data.

<b>Time Stamp Name</b>	Variable name of the time stamp. Time stamp data indicates the time at which step measurement was started.
<b>Index Name</b>	Variable name of the data index. Integer above 0 is returned.

Table 2-1

Available Type Values for Measurement Data Variables

Type	Description
SMU Quasi-Static C	SMU capacitance measurement data (F)
CMU Re(Z)	CMU resistance measurement data ( $\Omega$ )
CMU Im(Z)	CMU reactance measurement data ( $\Omega$ )
CMU Re(Y)	CMU conductance measurement data (S)
CMU IM(Y)	CMU susceptance measurement data (S)
CMU Frequency	CMU output signal frequency data (Hz)
CMU AC Level Output Data	CMU oscillator level output data (Vac)
CMU DC Bias Monitor Data	CMU DC bias monitor data (Vdc)
CMU AC Level Monitor Data	CMU oscillator level monitor data (Vac)

## Measurement Setup

This setup screen is used to define GPIB commands for controlling the SMU and CMU directly, and is used to set the SMU series resistor, the SMU filter, and the bias hold function.

### Advanced

This button opens a window for setting the SMU series resistor, the SMU filter, and the bias hold function. See “Advanced Setup” on page 2-34.

### Command Setup

This area defines GPIB commands for controlling the SMU and CMU directly.

**Command** Select the command or function from the pull-down menu. See “Command Setup” on page 2-36.

**Arguments** Enter the command parameter. To enter multiple parameters, insert a comma (,) between parameters. Multiple spaces can be inserted before and after the comma.

All parameters should be entered. If an optional parameter is abbreviated, the initial value may be set, the last used value may be reused, or no value may be set. The result is unique and depends on the command. For a description of each command, see *Agilent B1500 Programming Guide*.

The parameter can be a numeric variable or a numeric expression with fundamental arithmetic operations.

In an application test definition, the parameter can be a numeric type test parameters, device parameters, or Symbol.

If the internal variable is used, it must be used as a string variable. For example, the variables must be entered as follows.

“%I1”, “%R1”, “%R2”

**Ignore** Check this box to comment out this command line.

**Add** Adds a row for the command setup.

**Delete** Deletes the selected command setup. To select a command setup, use the radio button on the left.

**Up** Moves the selected command setup upward.

**Down** Moves the selected command setup downward.

## Advanced Setup

This window is opened by clicking the Advanced button on the Direct Control Measurement Setup screen, and is used to set the SMU series resistor, the SMU filter, and the bias hold function.

### Options

The following setup is available.

**Primary Setup** Primary setup ON or OFF.

If Primary Setup=ON, the primary setup is applied to the instrument when the Direct Control test is started. For the setup condition, see “Primary Setup” on page 2-35.

If Primary Setup=OFF, the primary setup is not applied. Set the Primary Setup to OFF if a setup exists and is proven to problem-free for running a test with. The test execution time can be reduced by this setting.

### Channel Settings

This area sets the series resistor and the filter.

**Unit** SMU used for the measurement

**VName** Variable name of voltage output or measurement data

**IName** Variable name of current output or measurement data

**Series R** Series resistor NONE or 1MOHM. See “SMU Series Resistor” on page 4-71.

**SMU Filter** SMU filter ON or OFF. See “SMU Filter” on page 4-70.

### After Measurement Setting

This area sets the bias hold function for maintaining the source output after measurement. The source applies the specified bias between two consecutive measurements of a repeated measurement or quick test. See “Bias Hold Function” on page 4-41.

**Bias Hold after Measurement** Bias hold function ON or OFF

### Semiconductor Relays (16440A SMU/PG Selector)

Effective only for the instrument installed with SPGU or PGU. Agilent 16445A selector adapter is needed for connecting Agilent 16440A SMU/PGU selector (B1500A-A04).

This area sets the selector CH1/CH3 status (*DEFAULT* or *PGU OPEN*). To use this function, set the *DEFAULT* setting to *Always SMU* or *Normally PGU (AUX)* by using the SMU/PG Selector tab screen on the Configuration window before starting the SPGU output.

**CH1** CH1 status, DEFAULT or PGU OPEN.



**CH3** CH3 status, DEFAULT or PGU OPEN.

CH1 represents channel 1 on the first selector and CH3 represents channel 1 on the second selector. See “SMU/PG Selector” on page 4-49 for details on the selector.

## Primary Setup

Primary setup is the initial setting set for when the Direct Control test is started. This setting is different from the initial setting set by the device clear or the \*RST command. The differences are listed below.

- Data output format  
FMT 13, 0 is set.
- Program memory  
Setup is not changed. The last used setting is retained.
- Internal variables  
Setup is not changed. The last used setting is retained.
- Measurement condition setup commands  
Setup is not changed for the following commands. The last used setting is retained.

MM

WV, WI, WSV, WSI, WNX

PV, PI, PWV, PWI

BDV

QSV, QSO

LGI, LGV, LSI, LSV, LSSI, LSSV

BGI, BGV, BSI, BSV, BSSI, BSSV

FC, WDCV

PDCV, PWDCV

WFC

WACV

MDCV

## Command Setup

To define the Command Setup, set a command or a function to the Command field, and enter the command parameters to the Argument field. Also, be aware of the following note.

- FMT command's first parameter must be 13.
- Use the Unit value of Channel Setup to specify the channel.
- Use the ReadDataBuffer function to read measurement data.
- Program memory number must be within the range of 1 to 1000.
- SCR command parameter cannot be abbreviated.
- VAR command's second parameter must be within the range of 1 to 40. Hence, internal variables %I1 to %I40 and %R1 to %R40 are available.

A setup example is shown in Table 2-2. This example uses SMU1 to perform voltage output and current spot measurement.

Table 2-2

Command Setup Example

Command	Arguments	Description
FMT	13, 0	Sets the data output format.
MM	1, SMU1:HP	Sets the measurement mode.
TSC	1	Enables the time data output.
AV	10, 1	Sets the number of averaging.
DV	SMU1:HP, 0, 1.5, 0.1	Forces the DC voltage.
CMM	SMU1:HP, 0	Sets the SMU operation mode.
RI	SMU1:HP, 0	Sets the current measurement range.
TSR		Resets the timer.
XE		Starts the measurement.
ReadDataBuffer	1	Reads the measurement data.

## ReadDataBuffer

This function reads the measurement data. The number of data blocks to be read is specified by the *count* parameter. If *count* is abbreviated, this function reads one data block.

**Syntax**                    `ReadDataBuffer [count]`

**Parameters**            *count*                    Number of data blocks or the number of terminators to be read.  
  
A data block is a group of data that ends with a terminator. For the data output format, see *Agilent B1500 Programming Guide*.

**Example**                    The following example performs three high speed spot measurements and then reads three measurement data blocks.

Command	Arguments	Description
TI	1, 0	Measurement channel 1 performs the high speed spot measurement.
TI	2, 0	Measurement channel 2 performs the high speed spot measurement.
TI	3, 0	Measurement channel 3 performs the high speed spot measurement.
ReadDataBuffer	3	Reads three measurement data blocks.

---

**NOTE**                    The data output buffer is cleared when the Direct Control test is started.

---



---

**NOTE**                    If a SMU performs the force side measurement operation and the source data output is enabled (by the FMT command), the SMU's source data is deleted and the SMU's measurement data is stored instead of the source data. The force side measurement operation is the SMU operation which performs the voltage force and voltage measurement or the current force and current measurement.

---

## compenReZ

This built-in function performs compensation on the specified impedance data and returns the real data for the compensation results.

This function is effective for the CMU measurement data compensation while the switching matrix is used with the CMU to extend the measurement terminals. Define the `compenReZ` and `compenImZ` functions as the user functions on the Function Setup screen, so that the compensation is performed while the Direct Control test is being executed. Also enter the formula to calculate the measurement parameters. An example definition is shown in Figure 2-2. For the equations, see “About Impedance Measurements” on page 4-22.

### Syntax

`compenReZ(frequency, Real, Imaginary)`

### Parameters

<i>frequency</i>	Measurement frequency. Hz. Numeric scalar or vector data.  For the C-f measurement data, <i>frequency</i> should be a vector variable that stores multiple frequency values.
<i>Real</i>	CMU Re(Z) measurement result data. $\Omega$ . Numeric vector data.  <i>Real</i> should be a vector variable that stores the impedance real data before compensation.
<i>Imaginary</i>	CMU Im(Z) measurement result data. $\Omega$ . Numeric vector data.  <i>Imaginary</i> should be a vector variable that stores the impedance imaginary data before compensation.

### Example

**Name** `Rcompen`

**Definition** `compenReZ(Freq, ReZ, ImZ)`

*Rcompen* returns the impedance real data for the compensation results.

## compenImZ

This built-in function performs compensation on the specified impedance data and returns the imaginary data for the compensation results.

This function is effective for the CMU measurement data compensation while the switching matrix is used with the CMU to extend the measurement terminals. Define the compenReZ and compenImZ functions as the user functions on the Function Setup screen, so that the compensation is performed while the Direct Control test is being executed. Also enter the formula to calculate the measurement parameters. An example definition is shown in Figure 2-2. For the equations, see “About Impedance Measurements” on page 4-22.

<b>Syntax</b>	<code>compenImZ(<i>frequency</i>, <i>Real</i>, <i>Imaginary</i>)</code>	
<b>Parameters</b>	<i>frequency</i>	Measurement frequency. Hz. Numeric scalar or vector data.  For the C-f measurement data, <i>frequency</i> should be a vector variable that stores multiple frequency values.
	<i>Real</i>	CMU Re(Z) measurement result data. $\Omega$ . Numeric vector data.  <i>Real</i> should be a vector variable that stores the impedance real data before compensation.
	<i>Imaginary</i>	CMU Im(Z) measurement result data. $\Omega$ . Numeric vector data.  <i>Imaginary</i> should be a vector variable that stores the impedance imaginary data before compensation.
<b>Example</b>	<b>Name</b>	Xcompen
	<b>Definition</b>	<code>compenImZ(Freq, ReZ, ImZ)</code>
		<i>Xcompen</i> returns the impedance imaginary data for the compensation results.

## compenReY

This built-in function performs compensation on the specified admittance data and returns the real data for the compensation results.

This function is effective for the CMU measurement data compensation while the switching matrix is used with the CMU to extend the measurement terminals. Define the compenReY and compenImY functions as the user functions on the Function Setup screen as shown in Figure 2-2, so that the compensation is performed while the Direct Control test is being executed. Also enter the formula to calculate the measurement parameters. For the equations, see “About Impedance Measurements” on page 4-22. Figure 2-2 defines the following formula to calculate capacitance.

$$C = \text{Im}Y / (2 \times \text{PI} \times \text{Freq})$$

### Syntax

`compenReY(frequency, Real, Imaginary)`

### Parameters

<b><i>frequency</i></b>	Measurement frequency. Hz. Numeric scalar or vector data.  For the C-f measurement data, <i>frequency</i> should be a vector variable that stores multiple frequency values.
<b><i>Real</i></b>	CMU Re(Y) measurement result data. S. Numeric vector data.  <i>Real</i> should be a vector variable that stores the admittance real data before compensation.
<b><i>Imaginary</i></b>	CMU Im(Y) measurement result data. S. Numeric vector data.  <i>Imaginary</i> should be a vector variable that stores the admittance imaginary data before compensation.

### Example

<b>Name</b>	Gcompen
<b>Definition</b>	compenReY(Freq, ReY, ImY)

*Gcompen* returns the admittance real data for the compensation results.

Figure 2-2

### Function Setup Example

			Add	Delete	Up	Down
Name:	Unit:	Definition:				
PI		3.1415				
ImY	ohm	compenImY(Freq,G,B)				
ReY	ohm	compenReY(Freq,G,B)				
C	F	ImY/(2*PI*Freq)				

## compenImY

This built-in function performs compensation on the specified admittance data and returns the imaginary data for the compensation results.

This function is effective for the CMU measurement data compensation while the switching matrix is used with the CMU to extend the measurement terminals. Define the compenReY and compenImY functions as the user functions on the Function Setup screen as shown in Figure 2-2, so that the compensation is performed while the Direct Control test is being executed. Also enter the formula to calculate the measurement parameters. For the equations, see “About Impedance Measurements” on page 4-22. Figure 2-2 defines the following formula to calculate capacitance.

$$C = \text{Im}Y / (2 \times \text{PI} \times \text{Freq})$$

### Syntax

`compenImY(frequency, Real, Imaginary)`

### Parameters

<b><i>frequency</i></b>	Measurement frequency. Hz. Numeric scalar or vector data. For the C-f measurement data, <i>frequency</i> should be a vector variable that stores multiple frequency values.
<b><i>Real</i></b>	CMU Re(Y) measurement result data. S. Numeric vector data. <i>Real</i> should be a vector variable that stores the admittance real data before compensation.
<b><i>Imaginary</i></b>	CMU Im(Y) measurement result data. S. Numeric vector data. <i>Imaginary</i> should be a vector variable that stores the admittance imaginary data before compensation.

### Example

**Name** Bcompen

**Definition** `compenImY(Freq, ReY, ImY)`

*Bcompen* returns the admittance imaginary data for the compensation results.

## Function Setup

This setup screen is displayed by clicking the Function Setup tab of the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling / C-V Sweep / Direct Control test setup screen, and is used to set the user function and the analysis function.

Calculation of the function is automatically performed after measurement. And the functions defined in the Display Setup screen will be displayed on the Data Display window. The user functions are available for the X-Y Graph Plot, List Display, and Parameters area. The analysis functions are available for the Parameters area.

### Apply

This button performs calculation of the definition in this screen. If this button is clicked when the Data Display window associated with this test has been opened, the display is also updated.

The associated Data Display window will be the Data Display window that displays the last measurement data or that is recalled at last.

### User Function, Analysis Function

The User/Analysis Function area sets the user/analysis functions effective for the test. Up to 20 functions can be set. The user function result is referable from the other user function defined in the following lines. The analysis function result is referable from the other analysis function defined in the following lines.

<b>Name</b>	Function name
<b>Unit</b>	Unit of the function
<b>Definition</b>	Definition of the function. The following identifiers can be used. <ul style="list-style-type: none"><li>• Variables for the measurement/output data used in this test</li><li>• Functions defined in the above lines</li><li>• Local variables passed from the application test that calls this test as a component</li><li>• Built-in functions and global variables</li><li>• Read-out functions (not available for the user functions)</li></ul> Also see Chapter 5, “Built-in Programming Tool.”
<b>Add, Delete</b>	Adds a function, or deletes the selected function. To select the function, use the left radio button.
<b>Up, Down</b>	Moves the selected function upward or downward.



---

## Auto Analysis Setup

This setup screen is displayed by clicking the Auto Analysis Setup tab of the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling / C-V Sweep / Direct Control test setup screen, and is used to set the automatic analysis function. When a measurement finishes, the function automatically draws lines, a marker, or both on the X-Y Graph of the Data Display window.

You can set up two lines and one marker for the automatic analysis function.

In the Line 1 and Line 2 areas, you can set up the lines to be drawn.

In the Marker area, you can set up the marker.

### Apply

This button performs calculation of the definition in this screen. If this button is clicked when the Data Display window associated with this test has been opened, the display is also updated.

The associated Data Display window will be the Data Display window that displays the last measurement data or that is recalled at last.

### Interpolation Mode

Check this box to enable the interpolation mode. You can position marker between measurement points.

### Line 1, Line 2

This area sets the line 1 or line 2 automatically displayed and moved to the specified position after the measurement.

- |                     |   |
|---------------------|---|
| <b>Enable</b>       | Check this box to enable the line.  |
| <b>Fix</b>          | Check this box to fix the line on the graph after the auto analysis is performed. Remove the check to make the line free. |
| <b>Type</b>         | Select one of the available line types listed below.  |
| <b>Axis</b>         | Axis effective for the line. Specify one from Y1 to Y8.   |
| <b>First Point</b>  | See the following description for the line you selected.  |
| <b>Second Point</b> | See the following description for the line you selected.  |

Available line types:

- “Normal line”: For drawing a line between *any two* points.
- “Gradient line”: For drawing a line through *any point* with a specified gradient.
- “Tangent line”: For drawing a tangent line to a *measurement* point.

## Classic Test Definition

### Auto Analysis Setup

- “Regression line”: For drawing a regression line by using the measurement data included in the area specified by *any two* points.

#### Marker

This area sets the marker automatically displayed and moved to the specified position after the measurement.

**Enable** Check this box to enable the marker.

**Condition** Enter a variable name and a condition expression to specify where you want the marker to appear.

**After** Check this box to enable the additional condition.

Enter a second variable and condition expression. This sets up a search start condition for finding specified point.

For example, you can specify the following expressions to search for a measurement point that satisfies the first condition after the second condition is satisfied.

Condition:[DGM]=[MAX(DGM)\*0.01] [x] After [DGM]=[MAX(DGM)]

#### Normal line

The First Point area and the Second Point area are used to specify the points the line passes through. The following two ways are available to specify the point.

- X-Y Coordinate

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- Data Condition

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

#### Gradient line

The First Point area is used to specify the point the line passes through. The following two ways are available to specify the point.

- X-Y Coordinate

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- Data Condition

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

In the Gradient field, enter a gradient value or expression.

**Tangent line**

The First Point area is used to specify the measurement point the tangent line passes through.

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

**Regression line**

The First Point area and the Second Point area are used to specify the range of the measurement data used for the regression calculation. The following two ways are available to specify the point.

- X-Y Coordinate

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- Data Condition

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

## Display Setup

This setup screen is displayed by clicking the Display Setup tab of the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling / C-V Sweep / Direct Control test setup screen or by checking the Define own Output Parameters box on the Test Definition window, and is used to set the test result output.

### X-Y Graph

Selects the data to be displayed in the X-Y Graph Plot area. The X and Y1 to Y8 axes can be set.

<b>Name</b>	Specifies the parameter for the axis.
<b>Sharing</b>	<p>Specifies the group for sharing the Y axis scale. Select from Group 1, Group 2, Group 3, Group 4, or None (no grouping).</p> <p>The Scale, Min, and Max values are shared by the Y axes in the same group. The scale set by Run Time Auto Scale, Auto Scale, Zoom in, and/or Zoom out are also shared.</p> <p>If another layer on the Data Display window contains an X-Y trace which has the same name for both its grouped Y parameter and its X parameter, the scaling will also be shared by this X-Y trace.</p>
<b>Scale</b>	Scale of the axis, Linear or Log
<b>Min</b>	Minimum value of the axis
<b>Max</b>	Maximum value of the axis
<b>Add</b>	Adds a parameter (or axis).
<b>Delete</b>	Deletes the parameter selected with the radio button on the left.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

### List Display, Parameters

Selects the data displayed in the List Display area or the Parameters area of the Data Display window. Up to 20 parameters can be set for each area.

<b>parameter</b>	Specifies the parameter for the data output.
<b>Add</b>	Adds a row for the parameter.
<b>Delete</b>	Deletes the selected parameter. To select the parameter, use the left radio button.

<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

## SMU Range Setup Window

This window is opened by clicking the Range button on the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling Measurement Setup screen, and is used to set the measurement ranging operation.

For the measurement ranging function, see “SMU Ranging Mode” on page 4-55.

<b>Unit</b>	SMU used for the measurement
<b>Name</b>	Variable name for the measurement data
<b>Mode</b>	Ranging mode, AUTO (auto ranging), LIMITED (limited auto ranging), or FIXED (fixed range)
<b>Range</b>	Range value. For auto ranging, ignore this field. For limited auto ranging, the measurement channel never uses ranges less than the specified range.

### Range Change

#### Rule and Rate

Not available for voltage measurement or fixed ranging. See also “Enhanced Auto Ranging for Current Measurement” on page 4-58.

Available *Range Change Rule* values are BY FULL RANGE, GO UP AHEAD, and UP AND DOWN AHEAD.

BY FULL RANGE performs normal auto ranging operation.

For the GO UP AHEAD and UP AND DOWN AHEAD rules, specify the *Rate* value, 11 to 100, which fixes the boundary of the ranging. The ranging occurs when the measurement data goes across the *boundary* values shown below.

$$\text{go up boundary} = \text{present measurement range} \times \text{Rate} / 100$$

$$\text{go down boundary} = \text{present measurement range} \times \text{Rate} / 1000$$

The *go down boundary* is available only for the UP AND DOWN AHEAD rule.

**Up** Moves the selected measurement channel setup upward.

**Down** Moves the selected measurement channel setup downward.

If multiple measurement channels are defined, the channels perform measurement sequentially. The order is top to bottom of the channels defined in this screen. For exceptions and details, see “Multiple Measurement Channels” on page 4-69.

---

**NOTE**

**To use 1 pA range, only for B1500A**

The measurement channel connected to the ASU (Atto Sense and Switch Unit) supports the 1 pA range. To use the 1 pA range, set the 1 pA fixed range or the 1 pA limited auto ranging.

The B1500A automatically performs the compensation of the data measured by the 1 pA range and returns the compensated data. You can use either the pre-stored offset data or the pre-measured offset data.

To measure the offset data, perform the self-calibration before starting the measurement for a DUT. This offset data is temporarily memorized until the B1500A is turned off.

---

## ADC and Integration Time Setup Window

This window is opened by clicking the ADC/Integ button on the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling / C-V Sweep Measurement Setup screen, and is used to set the A/D converter of the SMU measurement channels.

### A/D Converter

This area selects the ADC type and enables/disables the auto zero function. See “Integration Time” on page 4-65.

<b>Unit</b>	SMU used for the measurement
<b>Name</b>	Variable name for the measurement data
<b>ADC</b>	ADC type, HR ADC (high resolution ADC) or HS ADC (high speed ADC)  <i>HR ADC</i> is not available for MCSMU, HCSMU, and HVSMU. Also it is not available for the measurements using SMU pulse.
<b>Measurement</b>	Compliance Side only, or both Force and Compliance Sides  <i>Force and Compliance Sides</i> is not available for HRSMU, MPSMU, and HPSMU.

### Integration Time

This area sets the ADC mode and its factor for each ADC type.

<b>Mode</b>	ADC operation mode  AUTO, MANUAL, or PLC for High Resolution ADC AUTO, MANUAL, PLC, or TIME for High Speed ADC TIME or PLC for the measurements using SMU pulse.
<b>Factor</b>	Factor value. <i>Factor</i> value of “Integration Time” on page 4-65.
<b>Auto Zero</b>	Auto zero function ON or OFF. Only for the high resolution ADC. See “ADC Zero Function” on page 4-66.

### Pulse Averaging Count

This area sets the averaging of the pulsed measurement. Not effective for HRSMU, MPSMU, and HPSMU. The Count value is automatically set to the same value as the Count value on the Pulse Setup window for using multiple SMU pulse output channels.

<b>Count</b>	Number of pulsed measurements for averaging to get one measurement data.
--------------	--



---

## Pulse Setup Window

This window is opened by clicking the Pulse button on the Multi Channel I/V Sweep Measurement Setup screen, and is used to set the SMU pulse setup parameters. For the SMU pulse setup parameters, see “SMU Pulse” on page 4-62.

The followings are the setup parameters for pulse output.

### Common

The following parameters are common to the all pulse output channels.

**Period** Pulse period. Entering AUTO sets the optimum value automatically.

### Measurement

**Delay Time** Delay time from the beginning of the pulse period to the beginning of the measurement. Entering AUTO sets the optimum value automatically so that the measurement is completed when the transition from peak to base is started by the pulse output channel which starts the transition at first.

### Integration Time

The following entry fields are available for setting the high speed ADC. High resolution ADC is not available for this measurement.

**Mode** ADC operation mode, TIME or PLC

**Factor** Factor value. *Factor* value of “Integration Time” on page 4-65.

### Pulse Averaging Count

The following parameter sets the averaging of the pulsed measurement. The Count value works with the Count value on the ADC and Integration Time Setup window.

The Count value is not effective for HRSMU, MPSMU, and HPSMU.

**Count** Number of pulsed measurements for averaging to get one measurement data.

### Pulse Channels

The followings are the pulse setup parameters set to the pulse output channels individually.

**Unit** SMU used for the pulse output

**Name** Variable name for the pulse voltage output data or the pulse current output data

Classic Test Definition  
Pulse Setup Window

<b>Pulse Delay Time</b>	Delay time from the beginning of the pulse period to the beginning of the transition from base to peak. This value must be 0 for HRSMU, MPSMU, and HPSMU.
<b>Width</b>	Pulse width. This value must be the same for HRSMU, MPSMU, and HPSMU.
<b>Base</b>	Pulse base value

---

## Advanced Setup Window

This window is opened by clicking the Advanced button on the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling Measurement Setup screen, and is used to set the following functions of the measurement channel.

### Channel Settings

This area sets the series resistor and the filter.

<b>Unit</b>	SMU used for the measurement
<b>VName</b>	Variable name of voltage output or measurement data
<b>IName</b>	Variable name of current output or measurement data
<b>Series R</b>	Series resistor NONE or 1MOHM. See “SMU Series Resistor” on page 4-71.  SMU series resistor is not available for MCSMU, HCSMU, and HVSMU.
<b>SMU Filter</b>	SMU filter ON or OFF. See “SMU Filter” on page 4-70.
<b>V Compliance</b>	Only for UHCU, UHVU, or HVMCU. Limit value for the voltage sweep output. The output voltage of the measurement resource is automatically measured, and the measured voltage is compared with this limit value. And then, if it exceeds the limit, the measurement status is set to the compliance status, and the sweep measurement stops according to the setting in the Sweep [xxxx] status field on the Measurement Setup screen. Voltage range and setting resolution depend on the measurement resource.

### Wait Time Control

This area sets the source output wait time and the measurement wait time. See “Wait Time” on page 4-68. The Output Wait sets the source output wait time. The Measurement Wait sets the measurement wait time.

<b>Factor</b>	Factor value. 0 to 10, 0.1 resolution. This is the <i>N</i> value shown in “Wait Time” on page 4-68.
---------------	--

**After Measurement Settings** This area sets the bias hold function used to keep the source output after measurement. The source applies the specified bias between a measurement and the next measurement performed in the repeat measurement or quick test. See “Bias Hold Function” on page 4-41.

**Bias Hold after Measurement** Bias hold function ON or OFF

**Output Value after Measurement** Source output value after measurement  
START (sweep start value), STOP (sweep stop value),  
SOURCE (sampling output source value), or BASE (sampling output base value)

**Semiconductor Relays (16440A SMU/PG Selector)** Effective only for the instrument installed with SPGU or PGU. Agilent 16445A selector adapter is needed for connecting Agilent 16440A SMU/PGU selector (B1500A-A04).

This area sets the selector CH1/CH3 status (*DEFAULT* or *PGU OPEN*). To use this function, set the *DEFAULT* setting to *Always SMU* or *Normally PGU (AUX)* by using the SMU/PG Selector tab screen on the Configuration window before starting the SPGU output.

**CH1** CH1 status, *DEFAULT* or *PGU OPEN*.

**CH3** CH3 status, *DEFAULT* or *PGU OPEN*.

CH1 represents channel 1 on the first selector and CH3 represents channel 1 on the second selector. See “SMU/PG Selector” on page 4-49 for details on the selector.

## Range Setup Window

This window is opened by clicking the Range button on the C-V Sweep Measurement Setup screen, and is used to set the measurement ranging operation.

**CMU Measurement Range** This area sets the CMU measurement ranging operation.

<b>Unit</b>	CMU used for the measurement
<b>Mode</b>	Ranging mode, AUTO (auto ranging) or FIXED (fixed range). If Mode=AUTO, the MFCMU automatically searches for and measures at the range that provides the best resolution. If Mode=FIXED, the MFCMU measures at the range specified by the Range field.
<b>Range</b>	Measurement range. This field is effective when Mode=FIXED. Available measurement ranges depend on the measurement frequency setting. See Table 2-3.

**Table 2-3 Available Measurement Ranges**

Frequency	Available measurement ranges
≤ 200 kHz	50 Ω, 100 Ω, 300 Ω, 1 kΩ, 3 kΩ, 10 kΩ, 30 kΩ, 100 kΩ, 300 kΩ
≤ 2 MHz	50 Ω, 100 Ω, 300 Ω, 1 kΩ, 3 kΩ, 10 kΩ, 30 kΩ
≤ 5 MHz	50 Ω, 100 Ω, 300 Ω, 1 kΩ, 3 kΩ

**SMU Measurement Range** This area sets the SMU measurement ranging operation.

For the measurement ranging function, see “SMU Ranging Mode” on page 4-55.

<b>Unit</b>	SMU used for the measurement
<b>Name</b>	Variable name for the measurement data
<b>Mode</b>	Ranging mode, AUTO (auto ranging), LIMITED (limited auto ranging), or FIXED (fixed range)
<b>Range</b>	Range value. For auto ranging, ignore this field. For limited auto ranging, the measurement channel never uses ranges less than the specified range.

## Classic Test Definition Range Setup Window

### Range Change

#### Rule and Rate

Not available for voltage measurement or fixed ranging. See also “Enhanced Auto Ranging for Current Measurement” on page 4-58.

Available *Range Change Rule* values are BY FULL RANGE, GO UP AHEAD, and UP AND DOWN AHEAD.

BY FULL RANGE performs normal auto ranging operation.

For the GO UP AHEAD and UP AND DOWN AHEAD rules, specify the *Rate* value, 11 to 100, which fixes the boundary of the ranging. The ranging occurs when the measurement data goes across the *boundary* values shown below.

$$\text{go up boundary} = \text{present measurement range} \times \text{Rate} / 100$$

$$\text{go down boundary} = \text{present measurement range} \times \text{Rate} / 1000$$

The *go down boundary* is available only for the UP AND DOWN AHEAD rule.

#### Up

Moves the selected measurement channel setup upward.

#### Down

Moves the selected measurement channel setup downward.

If multiple measurement channels are defined, the channels perform measurement sequentially. The order is top to bottom of the channels defined in SMU Measurement Range. For exceptions and details, see “Multiple Measurement Channels” on page 4-69.

---

### NOTE

#### To use 1 pA range, only for B1500A

The measurement channel connected to the ASU (Atto Sense and Switch Unit) supports the 1 pA range. To use the 1 pA range, set the 1 pA fixed range or the 1 pA limited auto ranging.

The B1500A automatically performs the compensation of the data measured by the 1 pA range and returns the compensated data. You can use either the pre-stored offset data or the pre-measured offset data.

To measure the offset data, perform the self-calibration before starting the measurement for a DUT. This offset data is temporarily memorized until the B1500A is turned off.

---

---

## Advanced Setup Window for C-V Sweep

This window is opened by clicking the Advanced button on the C-V Sweep Measurement Setup screen, and is used to set the following functions.

### AC Level/DC Bias Monitor Settings

This area defines the variables for the AC level data and the DC bias data monitored by the CMU while the capacitance measurement is performed.

**Unit** CMU used for the measurement

#### AC Level Monitor

**Name** Variable name of the AC level monitor data

#### DC Bias Monitor

**Name** Variable name of the DC bias monitor data

### Channel Settings

This area sets the SMU series resistor and the SMU filter.

**Unit** SMU used for the measurement

**VName** Variable name of voltage output or measurement data

**IName** Variable name of current output or measurement data

**Series R** Series resistor NONE or 1MOHM. See “SMU Series Resistor” on page 4-71.

SMU series resistor is not available for MCSMU, HCSTMU, and HVSMU.

**SMU Filter** SMU filter ON or OFF. See “SMU Filter” on page 4-70.

### Wait Time Control

This area sets the source output wait time and the measurement wait time of SMU. See “Wait Time” on page 4-68. The Output Wait sets the source output wait time. The Measurement Wait sets the measurement wait time.

**Factor** Factor value. 0 to 10, 0.1 resolution. This is the *N* value shown in “Wait Time” on page 4-68.

Classic Test Definition  
Advanced Setup Window for C-V Sweep

**After Measurement Settings** This area sets the bias hold function used to keep the source output after measurement. The source applies the specified bias between a measurement and the next measurement performed in the repeat measurement or quick test. See “Bias Hold Function” on page 4-41.

**Bias Hold after Measurement** Bias hold function ON or OFF

**Output Value after Measurement** Source output value after measurement  
START (sweep start value) or STOP (sweep stop value)

**Semiconductor Relays (16440A SMU/PG Selector)** Effective only for the instrument installed with SPGU or PGU. Agilent 16445A selector adapter is needed for connecting Agilent 16440A SMU/PGU selector (B1500A-A04).

This area sets the selector CH1/CH3 status (*DEFAULT* or *PGU OPEN*). To use this function, set the *DEFAULT* setting to *Always SMU* or *Normally PGU (AUX)* by using the SMU/PG Selector tab screen on the Configuration window before starting the SPGU output.

**CH1** CH1 status, DEFAULT or PGU OPEN.

**CH3** CH3 status, DEFAULT or PGU OPEN.

CH1 represents channel 1 on the first selector and CH3 represents channel 1 on the second selector. See “SMU/PG Selector” on page 4-49 for details on the selector.



---

## Switching Matrix Control

On the EasyEXPERT main screen, click the Classic Test tab and select the Switching Matrix Control icon. The main screen displays the Switching Matrix Control setup screen. This setup screen is used to create the relay control setup for Agilent B2200A/B2201A/E5250A Switching Matrix.

To apply the relay control setup, click the Single button. Before doing this, you have to establish the GPIB connection to the switching matrix by using the Switching Matrix tab screen of the Configuration window. See “Switching Matrix” on page 1-67.

### Setup Name

Used to set the name of the relay control setup. The name must be the identification for the setup saved to a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is recorded in the test result data and is used to differentiate the data in the test record list area.

### Connection

This area is used to create the relay control setup for the switching matrix.

- |               |  |
|---------------|--|
| <b>Add</b>    | Adds a row for the relay connection setup.                                 |
| <b>Delete</b> | Deletes the setup row. To select the setup row, use the left radio button. |
| <b>Up</b>     | Moves the selected setup row upward.                                       |
| <b>Down</b>   | Moves the selected setup row downward.                                     |

The setup row provides the following entry fields.

- |                  |  |
|------------------|--|
| <b>Operation</b> | Relay control operation<br>Open All, Switch Setup, Close Range, Open Output, Open Range, or Open Input. See Table 2-4. |
| <b>Input</b>     | Switching matrix input port number or label<br>Available for the Switch Setup, Close Range, and Open Input operations. |

Classic Test Definition  
Switching Matrix Control

- Outputs** Switching matrix output channel number or label
- Four entry fields are available for the Switch Setup and Open Output operations to specify the output channels.
- Two entry fields are available for the Close Range and Open Range operations to specify the range of output channels.
- Rule** Relay connection rule
- BBM: Breaks the previous connection and then makes the new connection for the specified outputs.
- MBBR: Makes the new connection and then breaks the previous connection for the specified outputs.

Table 2-4

**Relay Control Operations**

<b>Operation</b>	<b>Description</b>
Open All	Opens all relays.
Switch Setup	Makes the connection from the specified input port to the specified output channels. Up to four output channels can be specified at once.
Close Range	Makes the connection from the specified input port to the specified output channels. They are between two channels given by the Outputs fields.
Open Output	Opens the relays connected to the specified output channels. Up to four output channels can be specified at once.
Open Range	Opens the relays connected to the specified output channels. They are between two channels given by the Outputs fields.
Open Input	Opens the relays connected to the specified input port.

---

## SPGU Control

On the EasyEXPERT main screen, click the Classic Test tab and select the SPGU Control icon. The main screen displays the SPGU control screen. See “SPGU Module” on page 4-27 for the function of the SPGU.

### Setup Name

Used to set the name of the SPGU control setup. The name must be able to identify a setup saved in a My Favorite group (preset group). Enter a unique name that is distinct from other setup.

The name is recorded in the execution result data and is used to differentiate the data in the test record list area.

### Channel Definition

Defines the SPGU channel setups.

<b>Unit</b>	Semiconductor pulse generator unit (SPGU) used for the voltage pulse output or the arbitrary linear waveform voltage output
<b>VName</b>	Name of voltage output data
<b>Mode</b>	Output mode VPULSE (voltage pulse output) or ALWG (arbitrary linear waveform voltage output).  All SPGU channels must have the same output mode.
<b>Add</b>	Adds a row for the channel setup.
<b>Delete</b>	Deletes the channel setup. To select the channel setup, use the left radio button.
<b>Up</b>	Moves the selected channel setup upward.
<b>Down</b>	Moves the selected channel setup downward.

### SPGU Setup

Provides the following buttons used to open the following setup window.

<b>Pulse/ALWG</b>	Opens “SPGU Pulse Setup Window” if Mode=VPULSE.  Opens “SPGU ALWG Setup Window” if Mode=ALWG.
<b>Advanced</b>	Opens “Advanced Setup Window for SPGU Control”.

## Classic Test Definition

### SPGU Control

**Progress Monitor** Provides the following check boxes used to set the Data Display window displayed while the SPGU channel output is applied.

**Graph: % of Progress** Check this box to display the Percent of Progress vs. Time plot.

**List: Time Data** Check this box to display the Time data list.

No data may be displayed on the Data Display window if the SPGU output time is too short. The progress monitor will be used for long duration outputs, for example when the duration is longer than 30 seconds.

## Advanced Setup Window for SPGU Control

This window is opened by clicking the Advanced button on the SPGU Control screen, and is used to set the following functions of the SPGU channel.

**After Measurement Setting** Sets the bias hold function used to decide the channel output after the SPGU output operation.

**Bias Hold after Measurement** Bias hold function ON or OFF

If the bias hold function is OFF, the channel stops the output immediately after the SPGU output operation.

If the function is ON, the channel maintains the base voltage output (in the VPULSE mode) or the initial voltage output (in the ALWG mode) between a SPGU output operation and the next operation performed in the repeat execution or quick test. Channel initialization occurs at the beginning of the next operation.

**Semiconductor Relays (16440A SMU/PG Selector)** Sets the CH1/CH3 status (*DEFAULT* or *PGU OPEN*) of Agilent 16440A SMU/PGU selector (B1500A-A04). To use this function, set the *DEFAULT* setting to *Always SMU* or *Normally PGU (AUX)* by using the SMU/PG Selector tab screen on the Configuration window before starting the SPGU output.

**CH1** CH1 status, DEFAULT or PGU OPEN.

**CH3** CH3 status, DEFAULT or PGU OPEN.

CH1 represents channel 1 on the first selector and CH3 represents channel 1 on the second selector. See “SMU/PG Selector” on page 4-49 for details on the selector.

Agilent 16445A selector adapter is needed for connecting the SMU/PGU selector.

## SPGU Pulse Setup Window

This window is opened by clicking the Pulse/ALWG button on the SPGU Control screen when Mode=VPULSE, or by clicking the SPGU Pulse Setup button on the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling Measurement Setup screen. It is used to set the SPGU pulse output. See Figure 2-3 for the pulse setup parameters.

### Global Settings

Provides the following entry field. The setting is effective for all channels.

**Period** Pulse period. 20 ns to 10 s, 10 ns resolution.

### Operation

Provides the following radio buttons used to set the SPGU channel output operation mode, free run, pulse count, or duration. Select one of the following buttons to set the operation mode. The setting is effective for all channels.

**FREE RUN** In this mode, the SPGU continues the pulse output until the measurement is completed or stopped.

This mode is not available for the SPGU Control classic test.

**PULSE COUNT** In this mode, the SPGU outputs the specified number of pulses. Specify a number within the range of 1 to 1000000.

**DURATION** In this mode, the SPGU continues the output until the specified time elapses. Specify a time within the range of 1  $\mu$ s to 31556926 s (1 year).

### Pulse Settings

**Unit** Semiconductor pulse generator unit (SPGU) used for the voltage pulse output

**VName** Name of voltage output data

**Type** 2-Level Pulse or 3-Level Pulse. If you choose 3-Level Pulse, the Additional Pulse fields become effective.

**Base, Peak** Pulse base and peak voltage. 0 to  $\pm$  40 V, 1 mV resolution.

**Delay** Delay time. 0 s to *Period*-20 ns, resolution 2.5 ns or 10 ns (pulse transition time > 8  $\mu$ s).

**Width** Pulse width. 10 ns to *Period*-10 ns, resolution 2.5 ns or 10 ns (pulse transition time > 8  $\mu$ s).

**Leading, Trailing** Pulse transition time. 8 ns to 400 ms, resolution 2 ns or 8 ns (pulse transition time > 8  $\mu$ s).

Classic Test Definition  
 SPGU Pulse Setup Window

**SPGU setup view order**

- Up** Moves the selected channel setup upward.
- Down** Moves the selected channel setup downward.

Changes are effective only for the display on the SPGU Pulse Setup window and not memorized. They are not applied to the Channel Setup.

**Load Z**

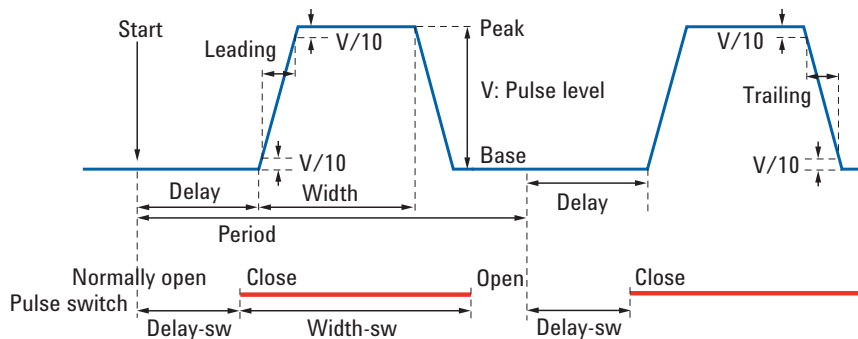
This button opens the “Load Z Setup Window” used to set the load impedance of the DUT (device under test) connected to the SPGU channel. The SPGU will automatically adjust the output voltage by using this value, and output the voltage close to the specified pulse base and peak values.

**Pulse Switch**

This button opens the “Pulse Switch Setup Window” used to set the pulse switch operation. The pulse switch is used to set the open condition of the SPGU output channel. The pulse switch is more durable than mechanical relays, and is better suited for frequent switching applications.

**Figure 2-3**

**SPGU Pulse Setup Parameters**



---

## Load Z Setup Window

This window is opened by clicking the Load Z button on the SPGU Pulse Setup window or the SPGU ALWG Setup window, and is used to set the load impedance of the DUT (device under test) connected to the specified SPGU channel.

Set the correct impedance value. The SPGU will automatically adjust the output voltage by using this value, and output the voltage close to the specified pulse base and peak values.

### Load Z

<b>Unit</b>	Semiconductor pulse generator unit (SPGU) used for the voltage pulse output or the arbitrary linear waveform voltage output
<b>VName</b>	Name of voltage output data
<b>Load Z</b>	Load impedance of the DUT (device under test) connected to this channel in $\Omega$ .

## Pulse Switch Setup Window

This window is opened by clicking the Pulse Switch button on the SPGU Pulse Setup window, and is used to set the pulse switch operation. The pulse switch is used to set the open condition of the SPGU output channel. The pulse switch is more durable than mechanical relays, and is better suited for frequent switching applications.

<b>Pulse Switch</b>	<b>Unit</b>	Semiconductor pulse generator unit (SPGU) used for the voltage pulse output
	<b>VName</b>	Name of voltage output data
	<b>SW Sync</b>	ENABLE (enables the pulse switch) or DISABLE (disables the pulse switch).  Switching of the pulse switch status (open and close) will be controlled automatically and synchronized with the output pulse, as shown in Figure 2-3.
	<b>Delay</b>	Delay time from start of pulse output to changeover of pulse switch. 0 s to <i>Period</i> -100 ns, 10 ns resolution.
	<b>Width</b>	Duration to hold the switched state of the pulse switch. 100 ns to <i>Period-Delay</i> ns, 10 ns resolution.
	<b>Normal</b>	CLOSE (normally closes the pulse switch) or OPEN (normally opens the pulse switch). The pulse switch will change the status after the Delay-sw time and keep the status until the Width-sw time elapses as shown in Figure 2-3. This operation is repeated every pulse period.



## SPGU ALWG Setup Window

This window is opened by clicking the Pulse/ALWG button on the SPGU Control screen when Mode=ALWG, or by clicking the SPGU ALWG Setup button on the I/V Sweep / Multi Channel I/V Sweep / I/V List Sweep / I/V-t Sampling Measurement Setup screen. It is used to set the arbitrary linear waveform output.

### Operation

Provides the following radio buttons used to set the SPGU channel output operation mode, free run, sequence count, or duration. Select one of the following buttons to set the operation mode. The setting is effective for all channels.

**FREE RUN** In this mode, the SPGU continues the ALWG sequence output until the measurement is completed or stopped.

This mode is not available for the SPGU Control classic test.

**SEQUENCE COUNT**

In this mode, the SPGU outputs the specified number of sequences. Specify a number within the range of 1 to 1000000.

**DURATION**

In this mode, the SPGU continues the output until the specified time elapses. Specify a time within the range of 1  $\mu$ s to 31556926 s (1 year).

### ALWG Setting

Provides the following field.

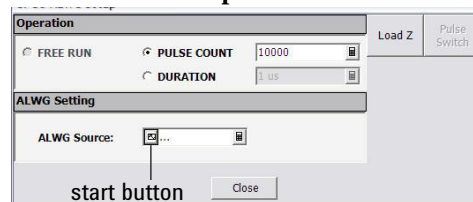
**ALWG Source** Click the start button in this field to open the Define ALWG Waveform window which is used to define the arbitrary linear waveform and its output sequence. See “Define ALWG Waveform Window” on page 2-68.

### Load Z

This button opens the “Load Z Setup Window” used to set the load impedance of the DUT (device under test) connected to the SPGU channel. The SPGU will automatically adjust the output voltage by using this value, and output the voltage close to the specified pulse base and peak values.

Figure 2-4

### SPGU ALWG Setup Window



## Define ALWG Waveform Window

This window is opened by clicking the start button in the waveform data entry field or the ALWG Source field on the SPGU ALWG Setup window, and is used to create the arbitrary linear waveform data which is defined by the pattern data and the sequence data.

### File menu

Provides the following five functions.

- Export ALWaveform...** Exports the waveform data.
- Import ALWaveform...** Imports the specified waveform data.
- Export Pattern...** Exports the pattern highlighted in the Patterns display area.
- Import Patterns...** Imports the specified pattern data.
- Close** Closes the Define ALWG Waveform window.

### Pattern menu

Provides the following six functions.

- New** This is for creating a new pattern. Opens the New Pattern dialog box. Entering a new name opens Waveform Pattern Editor (p. 2-71).
- Edit** Opens the Waveform Pattern Editor with the pattern highlighted in the Pattern display area.
- Cut** Cuts the pattern highlighted in the Pattern display area and stores it on the Clipboard.
- Copy** Copies the pattern highlighted in the Pattern display area to the Clipboard.
- Paste** Pastes the pattern stored on the Clipboard.
- Rename** Renames the pattern highlighted in the Pattern display area.

### Column menu

Provides the following five functions.

- Edit** Opens the Waveform Pattern Editor with the column data highlighted in the Sequence Editor area.
- Insert** Adds a column.

<b>Delete</b>	Deletes the column data highlighted in the Sequence Editor area.
<b>Move</b>	Moves the highlighted column to the left or the right.
<b>Repeat Count</b>	Opens the Repeat Count dialog box used to set the repeat count.

### Sequence Editor

Specifies the output channel, the output pattern, the repeat count of the pattern, and the output sequence of patterns. These setup information create the arbitrary linear waveform data.

<b>Column No.</b>	Number for specifying the output sequence. The pattern will be output in the order of this number.
<b>Repeat Count</b>	Output repeat count for the pattern specified by the <i>channel</i> field (e.g right field of the label “SPGU1:HV”). The repeat count must be entered in this field.

#### *channel*

(e.g SPGU1:HV) Output channel and its output pattern. The channel is specified by the header (e.g SPGU1:HV) of this row. The pattern data must be specified by this entry field. A pre-defined pattern can be selected from the drop down menu in this field. The right button opens the Waveform Pattern Editor.

This area provides the following buttons.

<b>Insert Column</b>	Adds a column.
<b>Delete Column</b>	Deletes the highlighted column.
<b>To Left</b>	Move the highlighted column to the left.
<b>To Right</b>	Move the highlighted column to the right.

This area also provides the following functions in the menu displayed by right-clicking this area.

<b>Cancel</b>	Closes the menu.
<b>Insert Column</b>	Adds a column.
<b>Delete Column</b>	Deletes the highlighted column.
<b>Move Column</b>	Moves the highlighted column to the left or the right.
<b>Repeat Count</b>	Opens the Repeat Count dialog box used to set the repeat count.
<b>Edit</b>	Opens the Waveform Pattern Editor with the specified pattern data (Pattern) or the specified column data (Column).

## Classic Test Definition

### Define ALWG Waveform Window

The Repeat Count field provides the following functions in the menu displayed by right-clicking the field.

- Cut** Cuts the highlighted value and stores it on the Clipboard.
- Copy** Copies the highlighted value to the Clipboard.
- Paste** Pastes the value stored on the Clipboard.

When a column is highlighted (light blue), the highlighted area provides the following functions in the menu displayed by right-clicking the field.

- Cancel** Closes the menu.
- Edit Column** Opens the Waveform Pattern Editor with the specified column data.

When a *channel* field is highlighted (light blue), the highlighted field provides the following functions in the menu displayed by right-clicking the field.

- Cancel** Closes the menu.
- Edit Pattern** Opens the Waveform Pattern Editor with the specified pattern data.

## Patterns

Displays icons for all pre-defined waveform patterns and provides the following functions in the menu displayed by right-clicking this area.

- Cancel** Closes the menu.
- New Pattern** This is for creating a new pattern. Opens the New Pattern dialog box. Entering a new name opens Waveform Pattern Editor (p. 2-71).
- Edit** Opens the Waveform Pattern Editor with the highlighted pattern.
- Cut** Cuts the highlighted pattern and stores it on the Clipboard.
- Copy** Copies the highlighted pattern to the Clipboard.
- Paste** Pastes the pattern stored on the Clipboard.
- Rename** Renames the highlighted pattern.
- Export Pattern...** Exports the highlighted pattern data.
- Import Pattern...** Imports the specified pattern data.

Double-clicking a pattern icon will open the Waveform Pattern Editor with the clicked pattern.

## Waveform Pattern Editor

This window is opened by several actions, for example by selecting New from the Pattern menu, and is used to create or edit the pattern data.

### File menu

Provides the following three functions.

<b>Save</b>	Saves the pattern data.
<b>Save As...</b>	Opens a dialog box used to specify a pattern name and save the pattern data.
<b>Close</b>	Closes the Waveform Pattern Editor without saving the data.

### Edit menu

Provides the following seven functions.

<b>Undo</b>	Undoes the last action.
<b>Redo</b>	Cancels the Undo operation.
<b>Add a New Point</b>	Adds a point.
<b>Delete a Point</b>	Deletes the highlighted point.
<b>Same Level As Left Point</b>	Sets the highlighted point to the same level as the left point.
<b>Copy Pattern to Clipboard</b>	Copies the highlighted pattern data to the Clipboard in the tab-separated spreadsheet format.
<b>Paste Pattern from Clipboard</b>	Pastes the tab-separated spreadsheet format data stored on the Clipboard.

### Display menu

Provides the following function.

<b>Channel</b>	Opens the Display Channels dialog box used to select the SPGU channels displayed on the Waveform Pattern Editor. This function is available if the Waveform Pattern Editor was opened with the column data.
----------------	---

## Pattern Editor

Defines the pattern data. The Pattern Editor provides two methods, graphic mode and tabular mode, for defining the pattern data.

In the graphic mode, you can define the pattern data by using the mouse pointer and GUIs on the graph (on the left side of the Pattern Editor).

In the tabular mode, you can define the pattern data by entering Time in seconds, Level in V, and Switch 0 (pulse switch open) or 1 (pulse switch close) into the table (on the right side of the Pattern Editor).

The Delta Time check box and the Delta Level check box are used to switch the data expression in the table. If the box is not checked, the value will be an absolute value; if the box is checked, the value will be the difference from the previous point.

The Pattern Editor provides the following buttons.

- Undo**                      Undoes the last action.
- Redo**                      Cancels the Undo operation.
- Add a New Point**      Adds a point.
- Delete a Point**        Deletes the highlighted point.

The Pattern Editor also provides the following functions in the menu displayed by right-clicking the area.

- Cancel**                    Closes the menu.
- Save**                      Saves the pattern data.
- Edit**                      Displays the Edit menu.
- Display**                  Displays the Display menu.

If the Pattern Editor is opened with the column data, the following GUI will be available.

- Ref Ch.**                    Specifies the channel or the pattern which contains the reference point.
- Ref Pt.**                    Specifies the reference point by the index number (No. column of table).
- Delta T**                    Displays the difference from the reference point. For time data.
- Delta V**                    Displays the difference from the reference point. For level data.

---

**3**

**Application Test Definition**

---

## Application Test Definition

This chapter provides the reference information of the application test definition setup GUI of Agilent EasyEXPERT software.

- “Test Definition Window”
- “Test Specification”
- “Test Output”
- “Test Contents”
- “Local Variable Definition”
- “Program Component”
- “Auto Analysis”
- “Data Display Control”
- “Display Data Setup”
- “GPIB I/O”
- “Message”
- “Data Store Control”
- “Command Execution”
- “Using Command Execution”



## Test Definition Window

The Test Definition window is opened by selecting the Define New Test... function or the Open Definition of This Test... function of the Library menu, and is used to create a test definition used in the application test mode.

The test definition consists of the elements shown in Figure 3-1. And the Test Definition window provides the following three tabs for creating a test definition.

- “Test Specification”

The test specification contains test name, description, device parameters, and test parameters. The parameters will be the variables used to pass the test conditions to the test execution flow or test setup defined in the test contents.

- “Test Contents”

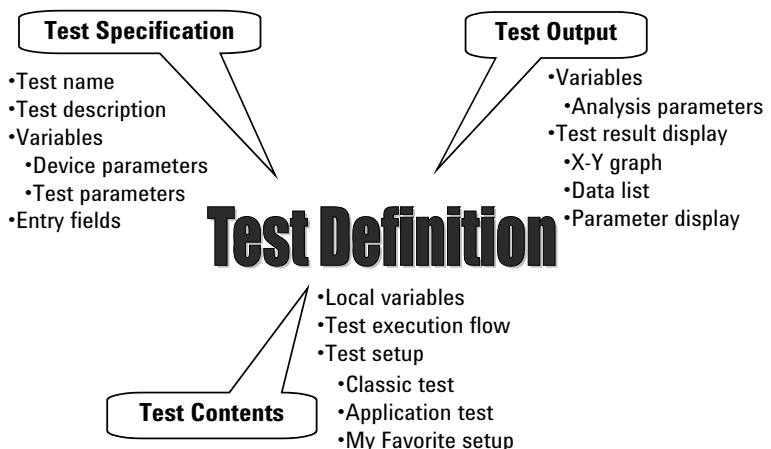
The test contents are the core of the test definition, and defines the local variables, test execution flow, and test setup. You can define the test setup by selecting application tests, classic tests, or My Favorite setup, and entering the test conditions.

- “Test Output”

The test output setup is optional. If you need the test result output (test result display and record) of the test definition, declare the variables (analysis parameters) and define the display setup.

Figure 3-1

### Test Definition



## Application Test Definition Test Definition Window

The Test Definition window provides the following menus.

### **File menu**

Provides the following two functions.

- |              |   |
|--------------|---|
| <b>Save</b>  | Saves the test definition. The sample application tests furnished with the EasyEXPERT and the application tests created by other users cannot be saved without changing the Test Name value. The Test Name must be your original.             |
| <b>Close</b> | Opens a dialog box that asks if you want to save before closing the window. Selecting Yes saves the test definition and closes the Test Definition window. Selecting No closes the Test Definition window without saving the test definition. |

### **Help menu**

**Agilent EasyEXPERT Help** Opens the EasyEXPERT help window.

---

## Test Specification

This setup screen is used to define the test definition information and the input parameters for the test definition.

### Test Information

The test information is defined in this area.

<b>Category</b>	Displays the category of the test definition.  The button opens a dialog box used to set the category. It must be 1 to 30 characters. Multiple categories can be set to a test definition.
<b>Test Name</b>	Specifies the test definition name. It must be 1 to 30 characters.
<b>Icon</b>	Specifies a bitmap file used for the test definition icon listed in the Library area of the main screen.  Browse...: Opens a dialog box used to select a bitmap file.  Cancel: Returns to the default setting.
<b>Description</b>	Enters the description of the test definition.

### Device Parameters Definition

The device parameters are defined in this area. Type of the device parameter is Numeric.

<b>Name</b>	Device parameter name
<b>Default</b>	Default value of the parameter
<b>Description</b>	Description of the parameter
<b>Add</b>	Adds a row for the device parameter.
<b>Delete</b>	Deletes the selected device parameter. To select the parameter, use the left radio button.
<b>Up</b>	Moves the selected device parameter upward.
<b>Down</b>	Moves the selected device parameter downward.

## Application Test Definition

### Test Specification

#### Test Parameters Definition

The test parameters are defined in this area.

<b>Background</b>	<p>Specifies a bitmap file used for the background in the Test Parameters area of the main screen.</p> <p>Browse...: Opens a dialog box used to select a bitmap file.</p> <p>Cancel: Returns to the default setting.</p>
<b>Name</b>	Test parameter name
<b>Type</b>	Type of the test parameter, Numeric, Module, String, Vector, or ALWaveform (SPGU ALWG output data)
<b>Default</b>	<p>Default value of the parameter</p> <p>For the Numeric, Vector, or ALWaveform type parameter, this field provides the button to open Numeric KeyPad at the right side.</p> <p>For the Vector type parameter, this field provides the button to open the “Define vector data (page 2-18)” dialog box at the left side.</p> <p>For the ALWaveform type parameter, this field provides the button to open the “Define ALWG Waveform Window (page 2-68)” at the left side.</p>
<b>Description</b>	Description of the parameter
<b>Align</b>	Specifies the X origin of the entry field in the main screen Test Parameters area. Check this box to set it to the left edge of the entry field. Uncheck this box to set it to the left edge of the parameter name placed to the left of the entry field.
<b>X, Y</b>	Specifies the entry field position. (X, Y)=(0, 0) specifies the upper left corner of the main screen Test Parameters area. Downward and rightward are positive. See “Layout...”.
<b>Width</b>	Specifies the width of the entry field.
<b>Ext</b>	<p>Do not check usually. The entry field for the parameter is put on the main screen.</p> <p>If the parameter is minor and do not need changes frequently, check the Ext check box. The entry field will be put on the dialog box displayed by clicking the “Extended Setup (page 1-30)” button.</p>
<b>Layout...</b>	Opens the “Define Layout (page 3-10)” dialog box.

<b>Add</b>	Adds a row for the test parameter.
<b>Delete</b>	Deletes the selected test parameter. To select the parameter, use the left radio button.
<b>Up</b>	Moves the selected test parameter upward.
<b>Down</b>	Moves the selected test parameter downward.
<b>Copy</b>	Copies the selected test parameter to the Clipboard.
<b>Paste</b>	Pastes the test parameter stored on the Clipboard.

## Properties

The Properties area is used to define the following values for the specified device parameter or test parameter. See “Device Parameters Definition” on page 3-5 and “Test Parameters Definition” on page 3-6.

To specify the parameter, select the radio button to the left of the parameter name.

<b>Min</b>	For Numeric or Vector parameters. Allowable minimum value.
<b>Max</b>	For Numeric or Vector parameters. Allowable maximum value.
<b>Digits</b>	For Numeric or Vector parameters. Number of effective digits.
<b>Resolution</b>	For Numeric or Vector parameters. Allowable setup resolution.
<b>Unit</b>	For Numeric or Vector parameters. Unit of the parameter.
<b>Typical Values...</b>	For Numeric or String parameters.  Opens the Define typical values dialog box used to define the selections (typical values) for the specified parameter.  The values defined here are the selections for the parameter entry field displayed on the main screen in the application test mode. The selections for the active entry field are displayed on the softkey. And one of the selections can be set by using the softkey. Also, it can be set by using the Select sub menu displayed by right-clicking the entry field.  The selections are effective for the specified parameter only.
<b>Resource Type</b>	For Module parameters.  Opens the Define Resource Types dialog box used to specify the type of measurement resource. Multiple resources can be set.

SMU, HPSMU, HCSMU, DHCSMU (dual HCSMU), HVSMU, HRSMU, MPSMU, MCSMU, HRSMU/ASU, GNDU, SPGU, HVSPGU, WGF MU, RSWGFMU, MFCMU, MFCMU/SCUU, SWM IN, SWM AUX IN, SWM SMU IN, UHCU500, UHCU1500, UHVU, or HVMCU

**Dimension...**

For Vector parameters.

Opens the Define dimensions dialog box for defining the dimension of vector data.

Use Dim1Min and Dim1Max to specify the primary dimension of vector data. This direction can be for the primary sweep data.

Use Dim2Min and Dim2Max to specify the secondary dimension of vector data. This direction can be for the secondary sweep data.

The following conditions must be satisfied.

- $1 \leq \text{Dim1Min} \leq \text{Dim1Max}$  (ex. Dim1Min=1, Dim1Max=11)
- $1 \leq \text{Dim2Min} \leq \text{Dim2Max}$  (ex. Dim2Min=1, Dim2Max=3)

**Assign Output Channels...**

For ALWaveform parameters.

This button opens the Assign ALWaveform Output Channels dialog box which specifies the variables if they are used to identify the SPGU channels. The variables must be the Module type test parameter.

For example, when the parameters are defined as shown below, PG1 should be set to the Assign ALWaveform Output Channels dialog box for the WAVE1 parameter.

PG1: Module parameter to identify the SPGU channel

WAVE1: ALWaveform parameter for the PG1 setup data

Use “Define ALWG Waveform Window (page 2-68)” to create the SPGU ALWG setup data. For the above example, the PG1 is the SPGU channel identifier when creating the WAVE1 data.

**Symbols...** For Numeric or Vector parameters.

Opens the Define dimensions dialog box for assigning a symbol (string value) to a numeric value for the specified numeric parameter.

By defining the numeric-symbol assignment, the symbol value, instead of the numeric value, is set to the parameter entry field displayed on the main screen in the application test mode.

In the test definition, the parameter works as a numeric value. However, the numeric-to-string translation result will not be the string value simply translated from the numeric value but the symbol assigned to the numeric value.

For example, if Symbol=ON is assigned to Value=1 of the Param1 parameter, the string ON will be set to the Param1 entry field when numeric value 1 is entered into the Param1 field. Also, while the Param1 field shows ON, the Param1 value will be 1 in the test definition. And when the numeric value of Param1 is 1, the numeric-to-string translation result will not be the string 1 but the string ON.

The numeric-symbol assignment is effective for the specified parameter only.

Value and symbol must be in a one-to-one relationship.

The value must satisfy the Min, Max, Digits, and Resolution settings. The symbol must be between 1 and 30 characters.

---

**NOTE**

**Symbols... numeric-symbol assignment**

For parameters defined with a numeric-string assignment, the numeric-to-string translation result will be the symbol assigned to the numeric value. Be careful when using the *string* built-in function that is used to translate a numeric expression to a string value.

For the following example setup of a numeric-symbol assignment, `string(Mode)` execution result at `Mode=0` is the string OFF, not the string 0.

Name=Mode, Value=0, Symbol=OFF

---

## Define Layout

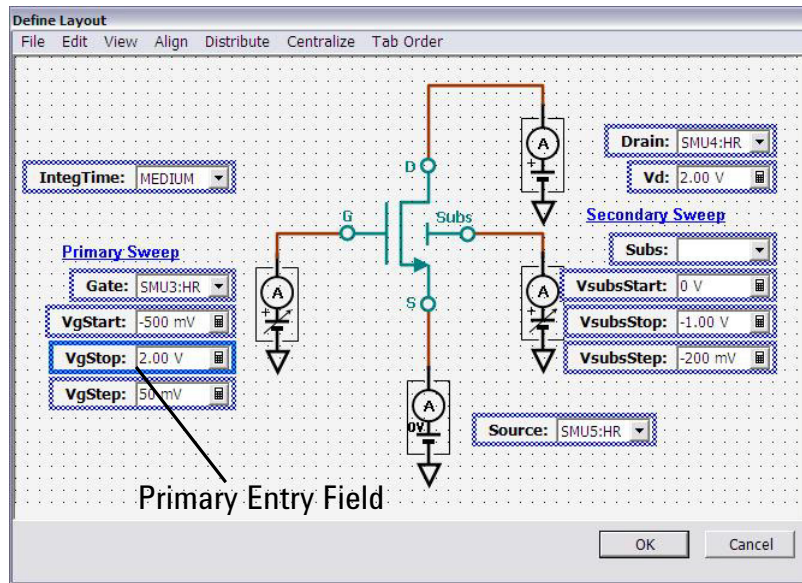
This dialog box is opened by clicking the Layout... button on the Test Specification screen of the Test Definition window, and is used to define the layout of the test parameter entry fields displayed on the application test setup screen. See Figure 3-2.

To define the layout of the entry field, follow the procedures below.

1. Select the entry fields to move. See “To Select Entry Fields” on page 3-13.
2. Move the selected entry fields. See “To Move Entry Fields” on page 3-13.

Figure 3-2

### Define Layout dialog box



## Define Layout GUI

This section describes the Graphical User Interface of the Define Layout dialog box.

The Define Layout dialog box provides the following buttons.

- |               |  |
|---------------|--|
| <b>OK</b>     | Saves the layout definitions and closes the Define Layout dialog box.      |
| <b>Cancel</b> | Closes the Define Layout dialog box without saving the layout definitions. |



The Define Layout dialog box provides the following menus.

<b>File menu</b>	<b>Exit</b>	Closes the Define Layout dialog box without saving the layout definitions.
<b>Edit menu</b>	<b>Undo</b>	Cancels an operation. Up to ten operations can be cancelled.
	<b>Redo</b>	Performs the canceled operation again.
	<b>Select All</b>	Selects all entry fields.
<b>View menu</b>	<b>Property...</b>	Opens the “Properties (page 3-12)” window.
<b>Align menu</b>	<b>Left</b>	Aligns the left ends of other selected entry fields to the left end of the primary entry field.
	<b>Center</b>	Aligns the horizontal centers of other selected entry fields to the horizontal center of the primary entry field.
	<b>Right</b>	Aligns the right ends of other selected entry fields to the right end of the primary entry field.
	<b>Top</b>	Aligns the upper ends of other entry fields to the upper end of the primary entry field.
	<b>Middle</b>	Aligns the vertical centers of other selected entry fields to the vertical center of the primary entry field.
	<b>Bottom</b>	Aligns the lower ends of other selected entry fields to the lower end of the primary entry field.
	<b>To Grid</b>	Aligns the upper left corner of the entry field to the nearest grid.
<b>Distribute menu</b>	<b>Horizontally</b>	Distributes the selected entry fields (3 or more) with even horizontal intervals. The leftmost entry field is used as the base point.
	<b>Vertically</b>	Distributes the selected entry fields (3 or more) with even vertical intervals. The leftmost entry field is used as the base point.
<b>Centralize menu</b>	<b>Horizontally</b>	Moves the horizontal center of all selected entry fields to the horizontal center of the entry field area.
	<b>Vertically</b>	Moves the vertical center of all selected entry fields to the vertical center of the entry field area.

<b>Tab Order menu</b>	<b>Top to Bottom</b>	Sets the TabIndex in ascending order of the X of the entry fields, starting at the upper left entry field. If the X of the entry fields are equal, the TabIndex is set in ascending order of the Y of the entry fields. For more information on X/Y, see “Properties” on page 3-12.
	<b>Left to Right</b>	Sets the TabIndex in ascending order of the Y of the entry fields, starting at the upper left entry field. If the Y of the entry fields are equal, the TabIndex is set in ascending order of the X of entry fields. For more information on X/Y, see “Properties” on page 3-12.

### Properties

This window is opened by clicking the Property... button in the View menu of the Define Layout dialog box, and specifies characteristics of the entry field area or the entry fields.

When the entry field area is selected, the following characteristics can be specified.

<b>Design</b>	<b>DrawGrid</b>	Sets whether to display the grid.
	<b>GridSize</b>	Sets the width and the height of the grid.
	<b>SnapToGrid</b>	Sets whether to snap to the grid.
<b>Layout</b>	<b>Size</b>	Sets the width and the height of the entry field area.
	When entry fields are selected, following characteristics can be specified.	
<b>Behavior</b>	<b>TabIndex</b>	Sets the tab traversing order for the test parameter entry fields of the application test.
<b>Layout</b>	<b>Align</b>	Specifies the X origin of the entry field. True: Left edge of the entry field False: Left edge of the parameter name placed to the left of the entry field
	<b>Width</b>	Specifies the width of the entry field.
	<b>X, Y</b>	Specifies the entry field position. The Y origin is the upper edge of the entry field.(X, Y)=(0, 0) specifies the upper left corner of the entry field area. Values increase downward and rightward.

## To Select Entry Fields

The following methods are available for selecting the entry fields.

- Click on the entry field. Multiple entry fields can be selected by clicking on the entry field while holding down the Ctrl key on the keyboard.
- Drag the mouse to draw a rectangle around multiple entry fields.
- Select Select All from the EDIT menu to select all entry fields.

The selected entry fields are outlined by the blue rectangles. The primary entry field, which is the base for layout operations, is surrounded by a highlighted blue rectangle, as shown in Figure 5-5. Only one primary entry field can be selected. To change the primary entry field, click on a selected entry field. The selected entry field that was most recently clicked will be the primary entry field.

The following methods are available for releasing (unselecting) the selected entry fields.

- Click the selected entry field while holding down the Shift key on the keyboard.
- Click outside of the selected entry fields to release all selected entry fields.

## To Move Entry Fields

The following methods are available for defining the layout of the selected entry fields.

- Drag & drop with the mouse to move the selected entry field(s).
- Change the X/Y values in the property window to move the selected entry field(s).
- Press the arrow keys on the keyboard to move the selected entry field(s).
- Using the functions in the Align menu, the selected single or multiple entry fields can be aligned to the primary entry field.
- Use the functions in the Distribute menu to evenly distribute the selected entry horizontally or vertically.
- Use the functions in the Centralize menu to move the horizontal or vertical center of all selected entry fields to the horizontal or vertical center of the entry field area.

## Test Output

This setup screen is used to set the test result output (test result display and record) that is the output of the test definition.

The test output setup is optional. If you have the following requirements, define the analysis parameters and set the display parameters.

- To send the test result data to the subsequent tests in an application test
- To make the program branching depends on the test result
- To make the calculation using the test data in the Test Contents
- To display/record the test result of this application test

To read the analysis parameters defined in the application test used in a test definition, use the “External Variable Setup (page 3-23)” dialog box.

**Define Own Output Parameters** Check this check box to set the test result outputs of the test definition. While this box is checked, the X-Y Graph, List Display, and Parameters areas are available. The test result output will be obtained every execution of the test definition.

**Allocate Data Display for each test** This field sets the Multi Display function to *Enable*, *Disable*, or *Comply with above level*.

If *Enable* is selected, the test result data of the same test setup name will be displayed on the exclusive Data Display window and the test result data of the different test setup name will be displayed on the new Data Display window.

If *Disable* is selected, the test result data will be always displayed on the singular Data Display window even if it is the test result data of the different test setup name.

Select *Comply with above level* for complying with the Multi Display ON/OFF status or the setting of the application test which uses this application test internally.

**Define Analysis Parameters...** This button opens the window. This “Analysis Parameter Definition (page 3-15)” window is used to define the vector data and the scalar data used in the test result output. The defined variable is called as the analysis parameter.

**X-Y Graph, List Display, Parameters** The areas are used to set the test result output (test result display and record) that is the output of the test definition. See “Display Setup” on page 2-46.

## Analysis Parameter Definition

This window is opened by clicking the Define Analysis Parameters button, and is used to define the vector variables and the scalar variables used in the test result output. The variables are called as the analysis parameters. This window is closed by the Close button.

---

### NOTE

Analysis parameters can be referred from a test definition which uses this application test. To read the value, make a mapping table between the analysis parameters and the local variables, device parameters, test parameters, or analysis parameters defined in the test definition by using the “External Variable Setup (page 3-23)” dialog box.

---

**Vector Parameters** This area sets the analysis parameters for the vector data. The vector data is an array data that contains numeric data more than one.

<b>Name</b>	Name of the analysis parameter
<b>Unit</b>	Unit of the analysis parameter
<b>Description</b>	Description of the analysis parameter
<b>Add</b>	Adds a row for the parameter.
<b>Delete</b>	Deletes the selected parameter. To select the parameter, use the left radio button.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

**Scalar Parameters** This area sets the analysis parameters for the scalar data. The scalar data is a numeric data.

<b>Name</b>	Name of the analysis parameter
<b>Unit</b>	Unit of the analysis parameter
<b>Description</b>	Description of the analysis parameter
<b>Add</b>	Adds a row for the parameter.
<b>Delete</b>	Deletes the selected parameter. To select the parameter, use the left radio button.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

## Test Contents

Test contents are the core of the test definition, and consists of the local variables, test setups, and test execution flow. In the test contents, the following elements can be defined. See Figure 3-3.

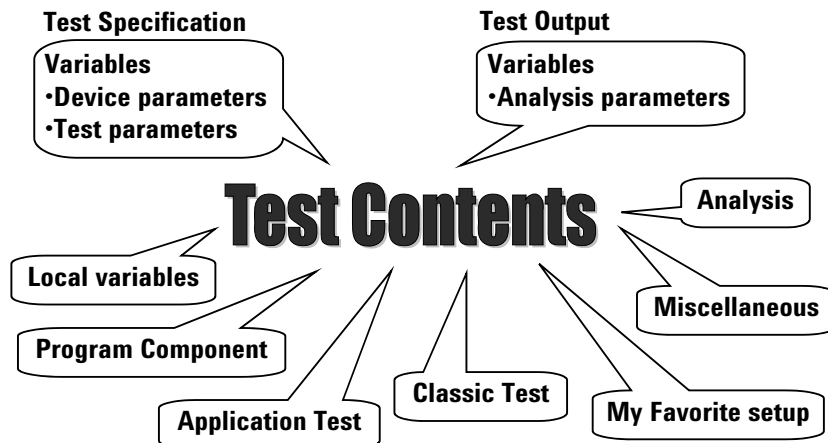
- Local variables
- Program components
- Application tests
- Classic tests
- My Favorite setup
- Analysis components
- Miscellaneous components
- Device parameters defined in Test Specification
- Test parameters defined in Test Specification
- Analysis parameters defined in Test Output

The device parameters, test parameters, and analysis parameters can be used in the Test Contents without defining as the local variables.

For the available components, see Figure 3-4.

Figure 3-3

### Test Contents



## Defining/Editing Test Contents

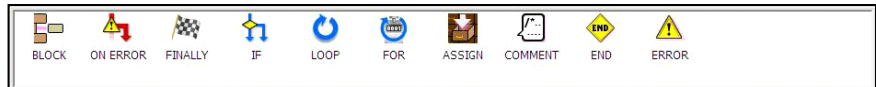
The Test Contents setup screen is used to define the test execution flow that is the core of the test definition. To define the test execution flow, use the test flow list area at the lower left of the window, and do following. Also see Figure 3-5.

1. Define local variables by using the Local Variables Definition statement.
2. Select a component from the component selection area. See Figure 3-4 for the available components.
3. Insert it between the BLOCK and END BLOCK statements.
4. Repeat this until your test execution flow is completed.

Figure 3-4

### Available Components

#### Program Component



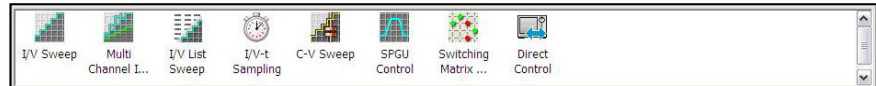
#### Application Test



#### Analysis



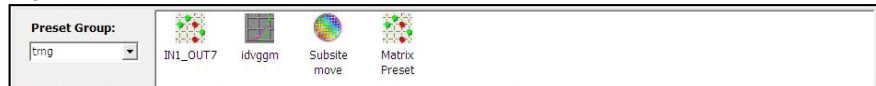
#### Classic Test



#### Miscellaneous



#### My Favorite



## Application Test Definition

### Test Contents

- Program Component** This component selection menu lists the program components available for the test flow. See “Program Component” on page 3-26.
- Application Test** This menu lists the application test setups available for the user. Inserting an application test setup to the test flow displays the test setup screen almost same as the “Application Test” on page 1-29. You set the test condition on this screen.
- Variables can be used for setting the test condition. To use a Numeric or Module variable, enter the variable name to the entry field. To use a String variable, type an equal sign (=) and enter the variable name (for example, =VAR).
- Clicking the Assign to external variables... button opens the External Variable Setup dialog box. See “External Variable Setup” on page 3-23.
- Analysis** This component selection menu lists the analysis functions available for the test flow, and provides the following components.
- “Auto Analysis” on page 3-27
  - “Display Data Setup” on page 3-31
  - “Data Display Control” on page 3-30
- Classic Test** This component selection menu lists the classic test objects, I/V Sweep, Multi Channel I/V Sweep, I/V List Sweep, I/V-t Sampling, C-V Sweep, SPGU Control Switching Matrix Control, and Direct Control.
- Inserting a classic test component to the test flow displays the test setup screen almost same as the “I/V Sweep” on page 2-4, “Multi Channel I/V Sweep” on page 2-9, “I/V List Sweep” on page 2-13, “I/V-t Sampling” on page 2-21, “C-V Sweep” on page 2-26, “SPGU Control” on page 2-61, “Switching Matrix Control” on page 2-59, or “Direct Control” on page 2-30. You set the test condition on the screen.
- Clicking the Assign to external variables... button opens the External Variable Setup dialog box. See “External Variable Setup” on page 3-23.
- My Favorite Setup** This component selection menu is used to select the My Favorite setup inserted to the test flow. Specify the preset group, then select the setup.
- If a tracer test setup is selected, it is converted to and inserted as a Multi Channel I/V Sweep test setup. In this conversion process, the VAR1 dual polarity sweep setup (Dual Polarity) is ignored. The setup of the positive sweep from start to stop is converted, but the setup of the negative sweep from –start to –stop is not converted.



## Miscellaneous

This component selection menu lists the miscellaneous functions available for the test flow, and provides the following components.

- “GPIB I/O” on page 3-33
- “Message” on page 3-38
- “Command Execution” on page 3-40
- “Data Store Control” on page 3-39

To add a component to the test flow, select the destination line for the component and drag the component from the component selection area to the test flow list area. Instead, you can select a line and a component, then click the Insert button. The component will be put on the line after the selected line, and the following lines will shift downward.

After adding a component to the test flow, selecting the new line might display an editor at the center to lower right of the window. The editor is used to define the setup. Some components provide the editor, and some components do not. For example, the Local Variables Definition provides the editor used to define the local variables. See “Local Variable Definition” on page 3-24.

You can edit the test flow by using the Block Selection/Line Selection, Insert, Delete, Copy, Cut, and Paste buttons. Also you can debug the test flow by using the Run/Pause, Abort, Step, Break, and Inspect buttons. See “Debugging Test Contents” on page 3-21.

## Edit / Debug

The Edit tab displays the buttons Block Selection/Line Selection, Insert, Delete, Copy, Cut, and Paste.

The Debug tab displays the buttons Run, Abort, Step, Break, and Inspect.

Only the buttons effective for the selected line in the list area are active.

## Block Selection / Line Selection

The Line Selection button changes the list area selection mode to the line selection mode. The Block Selection button changes the selection mode to the block selection.

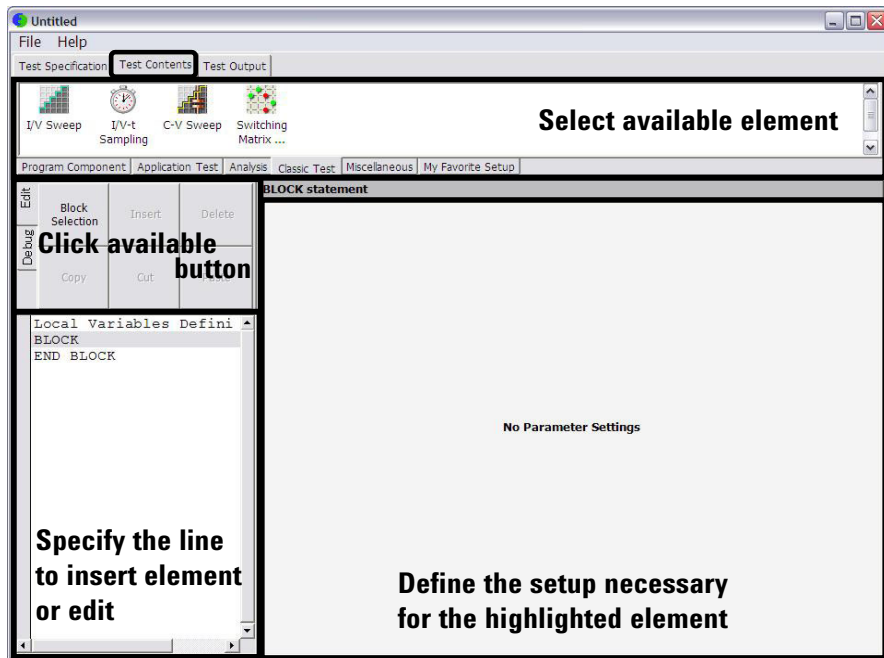
In the line selection mode, the line cursor selects every line.

In the block selection mode, the line cursor selects every block. For example, the lines from the IF statement to END IF statement can be a block.

## Application Test Definition Test Contents

- Insert** This button puts the selected component on the next line of the selected line. The following lines will shift downward.
- Delete** This button deletes the selected line. The following lines will shift upward.
- Copy** This button copies the selected line to the clipboard.
- Cut** This button moves the selected line to the clipboard. The following lines will shift upward.
- Paste** This button puts the components in the clipboard on the next line of the selected line. The following lines will shift downward.

Figure 3-5 Test Contents Screen

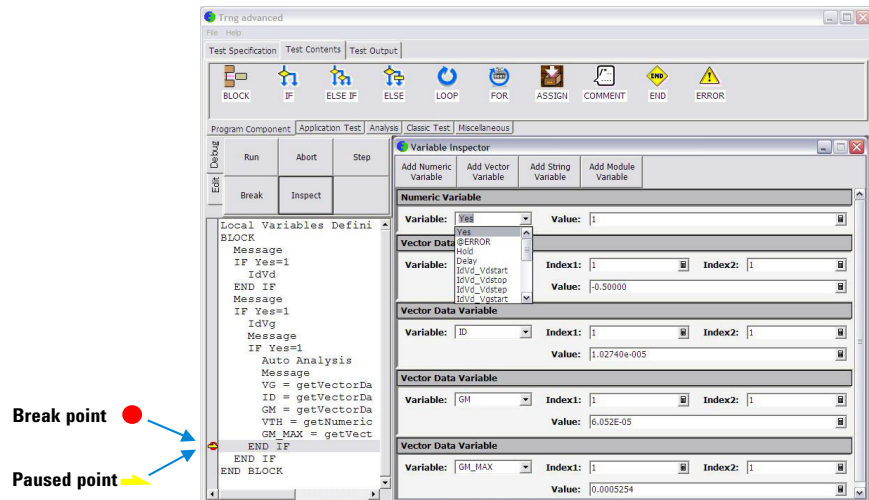


## Debugging Test Contents

On the Test Contents tab screen, click the Debug tab. The following buttons will appear. They are useful for debug of your test contents.

Set the break point, and click the Run button. And check the operation of the test contents, the value of variables by using the Variable Inspector, and so on.

**Figure 3-6** Debugging Test Contents



### Run/Pause

This button starts the debug (executes the test flow). During execution, the label changes to Pause. Clicking Pause pauses the execution, and changes the label to Run that is used to continue the debug (execution).

### Abort

This button aborts the debug (execution).

### Step

This button executes the selected line of the test flow. Clicking the button repeatedly continues the execution by a line.

### Break

This button sets/releases the break point. For the break point, this button releases the break point from the selected line. Program execution will break at the break point automatically.

### Inspect

This button is available when the debug (execution) is paused or broken. This button opens the Variable Inspector used to monitor value of device parameters, test parameters, analysis parameters, local variables, or system variables. See “Variable Inspector” on page 3-22.

## Variable Inspector

This Variable Inspector is displayed by clicking the “Inspect (page 3-21)” button on the Test Contents screen, and is used to monitor or change the value of the local variables, device parameters, test parameters, and analysis parameters. For the vector variables, only data monitor is available. The Variable Inspector provides the following buttons to add the variable monitor area.

**Add Numeric Variable** Adds the numeric variable monitor area.

**Add Vector Variable** Adds the vector variable monitor area.

**Add String Variable** Adds the string variable monitor area.

**Add Module Variable** Adds the module variable monitor area.

The following functions are available for the menu displayed by clicking the banner of the variable monitor area.

**Up** This button shifts the variable monitor area upward.

**Down** This button shifts the variable monitor area downward.

**Remove** This button deletes the variable monitor area.

**Numeric Variable** You can monitor or change the following information.

**Variable** Numeric variable name

**Value** Value of the variable

**Vector Variable** You can monitor the following information.

**Name** Vector variable name. Two dimensional array.

**Index1/2** Index for the primary/secondary plane of the two-dimensional array

**Value** Value of the variable

**String Variable** You can monitor or change the following information.

**Variable** String variable name

**Value** Value of the variable

**Module Variable** You can monitor or change the following information.

**Variable** Module variable name

**Value** Value of the variable

## External Variable Setup

This dialog box is displayed by clicking the Assign to external variable... button on an Application Test setup screen or a Classic Test setup screen, and is used to make a mapping table between the test setup internal variables and the local variables, device parameters, test parameters, or analysis parameters defined in the test definition which uses this test setup. After making the mapping table, you can read the value by using the external variables.

This dialog box provides the following buttons.

- Add** Adds the variable entry fields.
- Delete** Deletes the variable entry fields selected by the radio button.
- Up** Shifts the selected variable entry fields upward.
- Down** Shifts the selected variable entry fields downward.

This dialog box provides the following entry fields.

- External variable** Name of local variable, device parameter, test parameter, or analysis parameter connected with the variable set to the Measurement data field
- Measurement data** Name of internal variable used in a test setup and to be read. For the Application Test, only the analysis parameters are available.
- Example**

The following example connects the internal vector variable Vdata with the local variable VDATA.

  1. Define VDATA as a local variable by using the Local Variables Definition statement.
  2. Click the Assign to external variable... button on a test setup screen.
  3. Click the Add button on the External Variable Setup dialog box.
  4. Set VDATA to the External variable field.
  5. Set Vdata to the Measurement data field.
  6. Click the Close button.

## Local Variable Definition

This setup editor is displayed while the Local Variable Definition is selected in the test execution flow list area, and is used to define the variables used in the test definition. The setup editor provides the following buttons to add the variables.

**Add Numeric Variable** Adds the numeric variable.

**Add Vector Variable** Adds the vector variable.

**Add String Variable** Adds the string variable.

**Add Module Variable** Adds the module variable.

The following functions are available for the menu appeared by clicking the banner of the variable area.

**Up** This button shifts the variable area upward.

**Down** This button shifts the variable area downward.

**Remove** This button deletes the variable.

**Numeric Variable** This area provides the following entry fields.

**Name** Numeric variable name

**Value** Initial value of the variable

**Description** Additional information for the variable

### Vector Variable

This area provides the following entry fields.

**Name** Vector variable name. Two dimensional array.

**Count** Number of elements, or array size

**Description** Additional information for the variable

This area also provides the following radio button used to specify how to set the initial value of the array data automatically.

- Assign Linear Values FROM [*value of first element*] TO [*value of last element*]

If *Count* > 1, this sets  $first\ value + (N-1) \times (last\ value - first\ value) / (Count - 1)$  to the *N*-th element. If *Count* = 1, this sets *first value* (=last value) to the element.

- Assign Linear Values FROM [*value of first element*] STEP [*step value*]

This sets  $first\ value + (N-1) \times step$  to the *N*-th element.

### String Variable

This area provides the following entry fields.

**Name** String variable name

**Value** Initial value of the variable

**Description** Additional information for the variable

### Module Variable

This area provides the following entry fields.

**Name** Module variable name

**Description** Additional information for the variable

## Program Component

The Program Component selection menu provides the components described in “Script Program Statements” on page 5-20. Especially, the following components provide the setup editor.

- “ASSIGN”
- “COMMENT”
- “IF”
- “ELSE IF”
- “FOR”

### ASSIGN

This component provides the setup editor that contains the following entry fields used to define the value of the variable.

<b>Variable</b>	Variable name
<b>Expression</b>	Definition of the variable

### COMMENT

This component provides the setup editor used to enter the comment for the line.

### IF

This component provides the setup editor used to enter the condition for the IF statement.

### ELSE IF

This component provides the setup editor used to enter the condition for the ELSE IF statement.

### FOR

This component provides the setup editor that contains the following entry fields used to define the FOR NEXT loop condition.

<b>Variable</b>	Variable name
<b>Initial Value</b>	Count to start the FOR NEXT loop
<b>Terminate Value</b>	Count to terminate the FOR NEXT loop
<b>Step Value</b>	Loop count incremental step value



---

## Auto Analysis

This component provides the setup editor used to set up the automatic analysis function. When a measurement finishes, the function automatically draws lines, a marker, or both on the X-Y Graph of the Data Display window.

You can set up two lines and one marker for the automatic analysis function by using the setup editor.

In the Line 1 and Line 2 areas, you can set up the lines to be drawn.

In the Marker area, you can set up the marker.

**Interpolation Mode** Check this box to enable the interpolation mode. You can position marker between measurement points.

**Line 1, Line 2** This area sets the line 1 or line 2 automatically displayed and moved to the specified position after the measurement.

- Enable** Check this box to enable the line.
- Fix** Check this box to fix the line on the graph after the auto analysis is performed. Remove the check to make the line free.
- Type** Select one of the available line types listed below.
- Axis** Axis effective for the line. Specify one from Y1 to Y8.
- First Point** See the following description for the line you selected.
- Second Point** See the following description for the line you selected.

Available line types:

- “Normal line”: For drawing a line between *any two* points.
- “Gradient line”: For drawing a line through *any point* with a specified gradient.
- “Tangent line”: For drawing a tangent line to a *measurement* point.
- “Regression line”: For drawing a regression line by using the measurement data included in the area specified by *any two* points.

## Application Test Definition

### Auto Analysis

#### Marker

This area sets the marker automatically displayed and moved to the specified position after the measurement.

**Enable** Check this box to enable the marker.

**Condition** Enter a variable name and a condition expression to specify where you want the marker to appear.

**After** Check this box to enable the additional condition.

Enter a second variable and condition expression. This sets up a search start condition for finding specified point.

For example, you can specify the following expressions to search for a measurement point that satisfies the first condition after the second condition is satisfied.

Condition:[DGM]=[MAX(DGM)\*0.01] [x] After [DGM]=[MAX(DGM)]

#### Normal line

The First Point area and the Second Point area are used to specify the points the line passes through. The following two ways are available to specify the point.

- X-Y Coordinate

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- Data Condition

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

#### Gradient line

The First Point area is used to specify the point the line passes through. The following two ways are available to specify the point.

- X-Y Coordinate

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- Data Condition

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

In the Gradient field, enter a gradient value or expression.

### **Tangent line**

The First Point area is used to specify the measurement point the tangent line passes through.

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

### **Regression line**

The First Point area and the Second Point area are used to specify the range of the measurement data used for the regression calculation. The following two ways are available to specify the point.

- **X-Y Coordinate**

Enter the desired X-Y coordinate values or expressions in the X and Y fields.

- **Data Condition**

Enter a variable name and condition expression to specify the point. This is the same method as specifying the Marker point.

## Data Display Control

This component enables or disables the test result display of the application test or the classic test used in the test definition.

**Enable Data Display**    Checking this box enables the test result display.

If you do not use this component, the test result display is enabled.

This component cannot control the test result display of the test definition. The test result display is always enabled for the test definition. For the test result output of the test definition, see “Test Output” on page 3-14.

---

## Display Data Setup

This component is used to set the data output of the local variables defined in the test execution flow. The values can be displayed on the Data Display window.

### X-Y Graph

Selects the data to be displayed in the X-Y Graph Plot area. The X and Y1 to Y8 axes can be set.

<b>Name</b>	Specifies the parameter for the axis.
<b>Sharing</b>	<p>Specifies the group for sharing the Y axis scale. Select from Group 1, Group 2, Group 3, Group 4, or None (no grouping).</p> <p>The Scale, Min, and Max values are shared by the Y axes in the same group. The scale set by Run Time Auto Scale, Auto Scale, Zoom in, and/or Zoom out are also shared.</p> <p>If another layer on the Data Display window contains an X-Y trace which has the same name for both its grouped Y parameter and its X parameter, the scaling will also be shared by this X-Y trace.</p>
<b>Scale</b>	Scale of the axis, Linear or Log
<b>Min</b>	Minimum value of the axis
<b>Max</b>	Maximum value of the axis
<b>Add</b>	Adds a parameter (or axis).
<b>Delete</b>	Deletes the parameter selected with the radio button on the left.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

Application Test Definition  
Display Data Setup

**List Display,  
Parameters**

The List Display area sets the output for the List Display area of the Data Display window. The Parameters area sets the output for the Parameters area of the Data Display window. Up to 20 parameters can be set for each area.

<b>parameter</b>	Specifies the parameter for the data output.
<b>Add</b>	Adds a row for the parameter.
<b>Delete</b>	Deletes the selected parameter. To select the parameter, use the left radio button.
<b>Up</b>	Moves the selected parameter upward.
<b>Down</b>	Moves the selected parameter downward.

---

## GPIB I/O

This component provides the setup editor that contains the following entry fields. This component is used to control the GPIB interface. For the fields to enter a variable name, only the variable name defined in the test definition can be entered.

---

### CAUTION

Do not set the GPIB address of the B1500 to the Address parameter of this component even if this is on the Desktop EasyEXPERT software. Operation of the B1500 is not guaranteed.

---

### Write String

Sends the specified string data to the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>String</b>	String value to send
<b>Terminator</b>	Terminator, DEFAULT, EOI, CR/LF+EOI, LF+EOI, CR/LF, LF, or NONE (not specified)

### Write Value

Translates the specified numeric data to the string data, and sends it to the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>Format</b>	Format of Write Value. Follow Agilent T&M Programmers Toolkit.
<b>Value0</b>	Numeric value or string value to send
<b>Value1</b>	Numeric value or string value to send
<b>Value2</b>	Numeric value or string value to send
<b>Value3</b>	Numeric value or string value to send
<b>Terminator</b>	Terminator, DEFAULT, EOI, CR/LF+EOI, LF+EOI, CR/LF, LF, or NONE (not specified).

## Application Test Definition

### GPIB I/O

#### Read String

Reads a string value from the specified GPIB device.

**Address** GPIB address of the target device

**String** String variable name. Used to store the returned value.

**Length To Read** Byte length of the string value to read

**Length Actually Read** Numeric variable name. Used to store the byte length of the returned value. If the Length To Read is large enough and the terminator is sent with the returned string value, the value of this numeric variable is smaller than the specified Length To Read.

**Terminator** Terminator, DEFAULT, EOI, CR/LF+EOI, LF+EOI, CR/LF, LF, or NONE (not specified)

**Terminated** Numeric variable name. Used to store the execution result. If reading is completed by the terminator, 1 is entered. Otherwise, 0 is entered.

**Timeout** Timeout. The value must be less than  $2^{31}$  s, 1 ms resolution. 0 does not set timeout.

#### Read Value

Reads a numeric value from the specified GPIB device.

**Address** GPIB address of the target device

**Value** Numeric variable name. Used to store the returned value.

**Preamble** String variable name. Used to store preamble of the returned value. The preamble is a part considered as non-numeric value. If this field is not set, the preamble is deleted.

**Postamble** String variable name. Used to store postamble of the returned value. The postamble is a part considered as non-numeric value. If this field is not set, the postamble is deleted.

**Success** Numeric variable name. Used to store the execution result. For the successful result, 1 is entered. Otherwise, 0 is entered.

**Terminator** Terminator, DEFAULT, EOI, CR/LF+EOI, LF+EOI, CR/LF, LF, or NONE (not specified)

**Timeout** Timeout. The value must be less than  $2^{31}$  s, 1 ms resolution. 0 does not set timeout.



**Read List**

Reads a vector data from the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>Values</b>	Vector variable name. Used to store the returned value.
<b>Delimiter</b>	Delimiter. 1 character. 0 to 9, E, e, +, -, and . (comma) cannot be specified.
<b>Length Actually Read</b>	Numeric variable name. Used to store the number of data stored in the vector variable.
<b>Terminator</b>	Terminator, DEFAULT, EOI, CR/LF+EOI, LF+EOI, CR/LF, LF, or NONE (not specified)
<b>Timeout</b>	Timeout. The value must be less than $2^{31}$ s, 1 ms resolution. 0 does not set timeout.

**Read STB**

Reads the status byte of the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>Status Byte</b>	Numeric variable name. Used to store the status byte to read.
<b>Timeout</b>	Timeout. The value must be less than $2^{31}$ s, 1 ms resolution. 0 does not set timeout.

**GP-IB Control**

This area selects one from the following GPIB statements to execute.

- Group Execution Trigger

Sends a trigger to the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>Timeout</b>	Timeout. The value must be less than $2^{31}$ s, 1 ms resolution. 0 does not set timeout.

- Device Clear

Sends a device clear to the specified GPIB device.

<b>Address</b>	GPIB address of the target device
<b>Timeout</b>	Timeout. The value must be less than $2^{31}$ s, 1 ms resolution. 0 does not set timeout.

## Application Test Definition

### GPIB I/O

- Go To Local

Returns the specified GPIB device to the local state.

**Address** GPIB address of the target device

**Timeout** Timeout. The value must be less than  $2^{31}$  s, 1 ms resolution. 0 does not set timeout.

- Local LockOut

Locks the specified GPIB device.

**Address** GPIB address of the target device

**Timeout** Timeout. The value must be less than  $2^{31}$  s, 1 ms resolution. 0 does not set timeout.

#### Example 1

To send the following command to Agilent B2200A switching matrix (GPIB address 22), use Write Value function and enter the parameters as shown below.

```
:ROUT:SYMB:CHAN 0, 1, "Gate"
```

This command assigns the string Gate to the output 1 of the B2200A in the auto configuration mode.

- Write Value:

**Address** 22

**Format** :ROUT:SYMB:CHAN {0}, {1}, "{2}"

**Value0** 0

**Value1** 1

**Value2** Gate

**Value3** (blank)

**Terminator** LF+EOI

where, {0}, {1}, {2}, and {3} are the variables that can be used in the Format field, and corresponds to the Value0, Value1, Value2, and Value3 respectively.

**Example 2**

After performing Example 1, enter the following query command and read the response by using Write String and Read String functions as shown below. Use Write String to send the command, after that use Read String to read the response.

:ROUT:SYMB:CHAN? 0, 1

This command returns the string assigned to the output 1 of the B2200A in the auto configuration mode.

- Write String:

<b>Address</b>	22
<b>String</b>	:ROUT:SYMB:CHAN? 0, 1
<b>Terminator</b>	LF+EOI

- Read String:

<b>Address</b>	22
<b>String</b>	ReturnedValue
<b>Length To Read</b>	10
<b>Length Actually Read</b>	Length1
<b>Terminator</b>	LF+EOI
<b>Terminated</b>	Term
<b>Timeout</b>	1.00 s

where, ReturnedValue is a string variable to store the response of the :ROUT:SYMB:CHAN? 0, 1 command, Length1 is a numeric variable to store the byte length of the ReturnedValue value, and Term is a numeric variable to store the execution result. They must be defined in the Local Variables Definition statement.

## Message

This component provides the setup editor that contains the following entry fields. These are used to set up the message box that pauses program execution, appears on the screen, and waits for the response of the user.

<b>Message</b>	Enter the message displayed on the message box.
<b>User Response</b>	This area provides the Button 1, Button 2, Button 3, and Button 4 areas.
<b>Button 1</b>	This area provides the following entry fields. <b>Label</b> Label or name of the button placed at the left side <b>Selected</b> Variable name used in the test definition. The variable is used to store the response of the user. The returned value is 1 if the message box is closed by clicking this button. Otherwise, the returned value is 0.
<b>Button 2</b>	This area provides the following entry fields. <b>Label</b> Label or name of the button placed at the second from the left <b>Selected</b> Variable name used in the test definition. The variable is used to store the response of the user. The returned value is 1 if the message box is closed by clicking this button. Otherwise, the returned value is 0.
<b>Button 3</b>	This area provides the following entry fields. <b>Label</b> Label or name of the button placed at the second from the right <b>Selected</b> Variable name used in the test definition. The variable is used to store the response of the user. The returned value is 1 if the message box is closed by clicking this button. Otherwise, the returned value is 0.
<b>Button 4</b>	This area provides the following entry fields. <b>Label</b> Label or name of the button placed at the right side <b>Selected</b> Variable name used in the test definition. The variable is used to store the response of the user. The returned value is 1 if the message box is closed by clicking this button. Otherwise, the returned value is 0.

## Data Store Control

This component enables or disables the test record of the application test or the classic test used in the test definition.

### **Enable Creating Test Result Record**

Checking this box enables the test record.

If you do not use this component, the test record is enabled.

This component cannot control the test record of the test definition. The test record is always enabled for the test definition. For the test result output of the test definition, see “Test Output” on page 3-14.

## Command Execution

This component provides the setup editor that contains the following entry fields. This component is used to execute the specified execution file (EXE file) and obtain the execution results. For the fields to enter a variable name, only the variable name defined in the test definition can be entered. See “Using Command Execution” on page 3-42.

**Command  
Filename**

File name of the Windows execution file (EXE file) to execute.

**Argument**

Argument or parameter to pass to the EXE file.

**Write Type**

Type of data to write. String (string data) or List (vector data).

- String

Displays the Write String area used to specify the string data to write. Up to 20 string data can be defined.

- List

Displays the Write List area that provides the following entry fields.

**Values** Variable name of the List data to be passed to the EXE file.

**Format**

If you do not specify Format, this function passes the List data to the EXE file. To pass a data in the List data, enter the keyword to specify the element. See “Defining Format Field” on page 3-45.

## Read Type

Type of data to read. String (string data), Value (numeric data), List (vector data), or None (no response).

- String

Displays the Read String area that provides the following entry fields.

**Result** Numeric variable name. Used to store the execution result.

**String** String variable name. Used to store the returned value.

**Length**

**Actually Read** Numeric variable name. Used to store the byte length of the returned value.

- Value

Displays the Read Value area that provides the following entry fields.

**Result** Numeric variable name. Used to store the execution result.

**Value** Numeric variable name. Used to store the returned value.

- List

Displays the Read List area that provides the following entry fields.

**Result** Numeric variable name. Used to store the execution result.

**Values** Vector variable name. Used to store the returned value.

- None

This type is for the EXE file without response or data to read.

## Using Command Execution

Windows execution file (.exe file) can be executed by using the Command Execution component. Most execution files will need parameters to be sent or returned. The parameters will be variables that can be defined on the Test Specification tab screen or the Local Variable Definition component on the Test Contents tab screen.

This section describes how to enter the windows execution file.

- “Command Parameters”
- “Defining Numeric/Vector Input Parameter”
- “Defining String/Numeric Input Parameters”
- “Defining Numeric Output Parameter”
- “Defining Vector Output Parameter”
- “Defining String Output Parameter”
- “Defining Format Field”

### Command Parameters

The Windows Command Execution component supports one input variable and one output variable. To use multiple string input parameters or multiple numeric input parameters, set the variable as shown below. Multiple vector input parameters and multiple output parameters are not supported.

- If the command requires multiple string input parameters, use string variables. In the Windows Command Execution component, set the Write Type to String. Up to 20 string data can be defined.
- If the command requires multiple numeric input parameters, use numeric variables and set the Write Type to String. To enter the numeric variable name, use the *string* built-in function. For example, enter `string(Rz)` to set the Rz numeric variable.
- If the command requires both numeric and string input parameters, use numeric and string variables respectively, and set the Write Type to String.

---

**NOTE**

Before calling the command, enter the value to the input variable by using the ASSIGN statement.

---



---

**NOTE**

**string built-in function**

The *string* built-in function is used to translate a numeric expression to a string value. Be careful about the argument for this function. If it is a numeric variable defined with the numeric-symbol assignment, the *string* function execution result will be the symbol assigned to the numeric value, not the string value just translated from the numeric value. The numeric-symbol assignment is defined in the “Test Specification” on page 3-5.

For the following example setup of the numeric-symbol assignment, `string(Mode)` execution result at `Mode=0` is the string `OFF`, not the string `0`.

Parameter `Mode`, `Value=0`, `Symbol=OFF`

---

## Defining Numeric/Vector Input Parameter

1. Define a vector variable in the Test Specification tab screen or the Local Variable Definition component of the Test Contents tab screen.
2. Set the Write Type to List in the Windows Command Execution component.
3. Enter the variable name in the Values field of the Write List area.

For the numeric parameter, a data element of vector variable is used to store the data for the parameter. Specify the data by using the Format field. For example, enter `{ 0 }` to specify the data in the first element of the vector variable. For the Format field, see “Defining Format Field” on page 3-45.

For the vector parameter, ignore the Format field.

## Defining String/Numeric Input Parameters

1. Define string or numeric variables in the Test Specification tab screen or the Local Variable Definition component in the Test Contents tab screen.
2. Set the Write Type to String in the Windows Command Execution component.
3. Add a line in the Write String area.
4. Enter the variable name for the first input parameter in the first line.

If the command needs multiple input parameters, add a line and enter the variable name for the variable in the added line. Then observe the order of parameters.

To enter a numeric variable name, use the *string* built-in function. For example, enter `string(Rz)` to set the `Rz` numeric variable.

## Defining Numeric Output Parameter

1. Define a numeric variable in the Test Specification tab screen or the Local Variable Definition component of the Test Contents tab screen.
2. Set the Read Type to Value in the Windows Command Execution component.
3. Enter the variable name in the Value field of the Read Value area.

The Result field can be blank or the name of another numeric variable defined in the Test Specification or Local Variable Definition. The variable will be used to receive the command execution result.

## Defining Vector Output Parameter

1. Define a vector variable in the Test Specification tab screen or the Local Variable Definition component of the Test Contents tab screen.
2. Set the Read Type to List in the Windows Command Execution component.
3. Enter the variable name in the Values field of the Read List area.

The Result field can be blank or the name of a numeric variable defined in the Test Specification or Local Variable Definition. The variable will be used to receive the command execution result.

## Defining String Output Parameter

1. Define a string variable in the Test Specification tab screen or the Local Variable Definition component of the Test Contents tab screen.
2. Set the Read Type to String in the Windows Command Execution component.
3. Enter the variable name in the String field of the Read String area.

The Result field and the Length Actually Read field can be blank or the name of a numeric variable defined in the Test Specification or Local Variable Definition. The variable will be used to receive the command execution result or the length of the returned string data.

## Defining Format Field

The Format field requires a keyword to specify a data in vector variable. The keyword must have the style such as {I,A:F}. where,

- I** Index used to specify the element. Integer, 0 or positive number. Mandatory. 0 specifies the first element.
- A** Character length of the specified data element. Positive integer for right-aligned, or negative integer for left-aligned. If this value is not set, the length is not specified (no limit).
- F** Style of the data. See Table 3-1. If this value is not set, G is selected..

Followings are the example formats.

- {0,5:E} First element, five characters, exponential notation
- {1,10} Second element, 10 characters, general
- {2:x} Third element, no limit for length, hexadecimal numeric
- {3} Fourth element, no limit for length, general

**Table 3-1**

**Available F values**

<b>F</b>	<b>Meaning</b>	<b>F</b>	<b>Meaning</b>
C or c	Circulation	N or n	Numeric
D or d	Decimal numeric	P or p	Percent
E or e	Exponential notation	R or r	Round trip
F or f	Fixed point	X or x	Hexadecimal numeric
G or g	General		

## Setup Example

The following example setup is used to execute a `PlsdivMeasureId` command that has two numeric inputs and four numeric outputs.

*ranging* and *id\_range* are numeric variables used for the input parameters, *read\_data* is a vector variable used for the output parameters, and *PassFail* is a numeric variable used to store the command execution result. They must be defined in the Test Specification tab screen or the Local Variable Definition component on the Test Contents tab screen. Input parameter values must be set to the variables before executing the command. Four data will be assigned to *read\_data* in sequence.

Figure 3-7

### Windows Command Execution Setup Example

The screenshot shows the 'Windows Command Execution' dialog box. It is divided into several sections:

- Command Filename:** A text field containing the path `Program Files\Agilent\PLSDIV\PLSDIVExe\TISExe\PlsdivMeasureId.exe` and a 'Browse...' button.
- Argument:** An empty text field.
- Write Type:** Radio buttons for 'String' (selected) and 'List'.
- Read Type:** Radio buttons for 'String', 'Value', 'List' (selected), and 'None'.
- Write String:** A section with 'Add', 'Delete', 'Up', and 'Down' buttons. It contains two entries:
  - Radio button: `string(ranging)`
  - Radio button (selected): `string(id_range)`
- Read List:** A section with two dropdown menus:
  - Result:** Set to `PassFail`.
  - Values:** Set to `read_data`.

---

**4**

**Function Details**

---

## Function Details

This chapter explains the following measurement functions of Agilent B1500 and EasyEXPERT.

- “I/V Sweep Measurement”
- “Multi Channel I/V Sweep Measurement”
- “I/V-t Sampling Measurement”
- “C-V Sweep Measurement”
- “SPGU Module”
- “Sweep Abort Function”
- “Standby Function”
- “Bias Hold Function”
- “Current Offset Cancel”
- “SMU CMU Unify Unit”
- “Atto Sense and Switch Unit”
- “SMU/PG Selector”
- “Module Selector”
- “Ultra High Current Expander/Fixture”
- “HVSMU Current Expander”
- “Ultra High Voltage Expander”
- “SMU Ranging Mode”
- “SMU Compliance”
- “SMU Pulse”
- “SMU Measurement Time”
- “SMU Filter”
- “SMU Series Resistor”
- “Interlock Function”
- “Auto Power Off Function”
- “Initial Settings”

## I/V Sweep Measurement

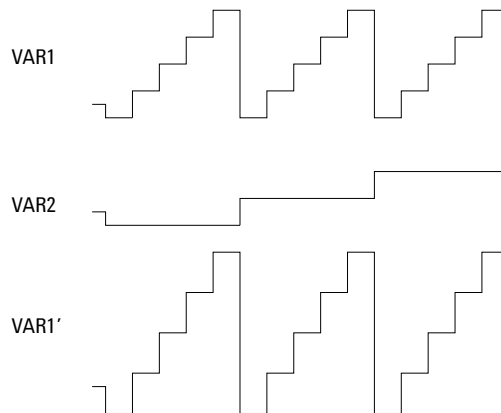
For the I/V sweep measurements, sweep source channels (SMU) perform staircase sweep output of voltage or current, while monitor channels (SMU) measure voltage or current for each sweep step. According to the output function, the following measurements are available. See Figure 4-1 for the SMU output function.

- “Basic Sweep Measurement” by using VAR1 function
- “Subordinate Sweep Measurement” by using VAR1 and VAR2 functions
- “Synchronous Sweep Measurement” by using VAR1 and VAR1’ functions, VAR2 function is also available
- “Pulsed Sweep Measurement”

This is the basic, subordinate, or synchronous sweep measurement using a pulse output SMU. Only one pulse output SMU is available.

Figure 4-1

### SMU Output Function and Sweep Output



### Available Measurement Resources

SMUs and SPGUs can be used. SMU can be constant voltage/current source or pulse voltage/current source and perform voltage/current measurement. SPGU can be pulse voltage source. See “SPGU Module” on page 4-27 for using the SPGU.

If multiple measurement channels are used, the channels perform measurement in order. See “Multiple Measurement Channels” on page 4-69.

## Output Range

Output range of the sweep source channel is automatically set to the following range.

- minimum range covers all sweep output values in Linear sweep.
- minimum range covers all sweep output values in Log voltage sweep.
- optimum range for output value every sweep step in Log current sweep.

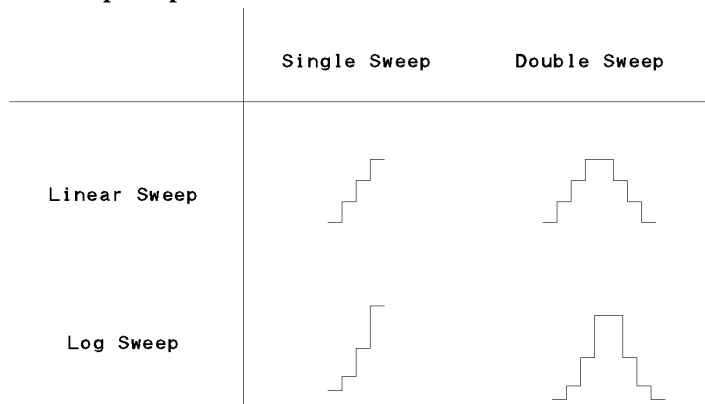
## Basic Sweep Measurement

Basic sweep measurement uses the primary sweep source (VAR1). You can select any combination of Direction and Linear/Log as shown in Figure 4-2.

- Direction
  - Single source channel sweeps the output from user specified *start* value to *stop* value.
  - Double source channel sweeps the output from user specified *start* value to *stop* value, then from *stop* value to *start* value.
- Linear/Log
  - Linear staircase
  - Logarithmic staircase

Figure 4-2

### Staircase Sweep Outputs



See “I/V Sweep” on page 2-4 for the GUI.



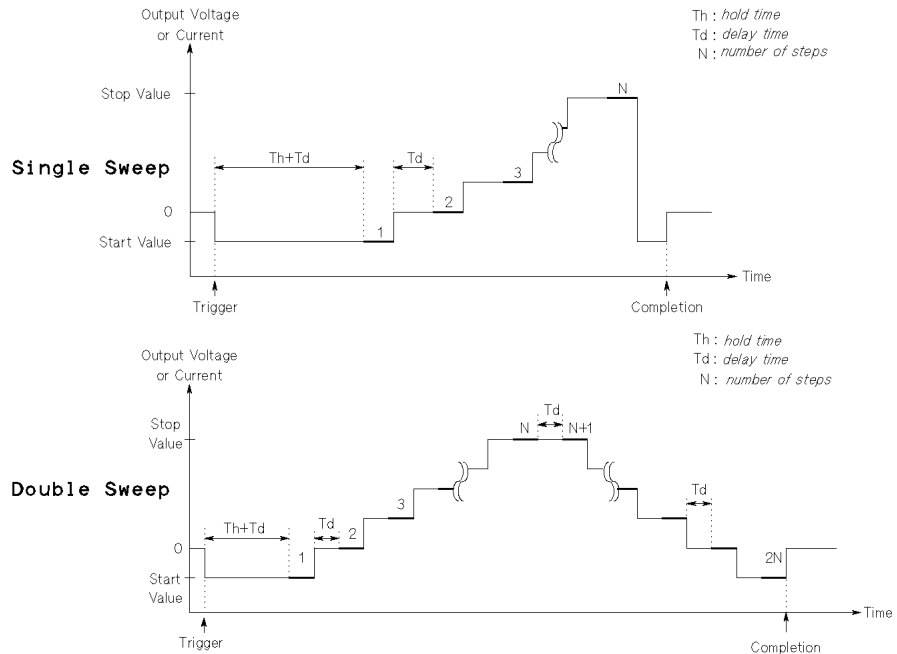
The following setup is required to perform the basic sweep measurements.

1. Select the following on the Channel Setup tab screen.
  - Select VAR1 for the Function of the SMU used for VAR1.
  - Select V, VPULSE, or COMMON for the Mode of current monitor SMU, or I or IPULSE for voltage monitor SMU. Pulse is available only for one SMU.
2. Specify the following parameters for VAR1 on the Measurement Setup tab screen.

<b><i>Direction</i></b>	Single or double sweep.
<b><i>Linear/Log</i></b>	Linear or logarithmic sweep. For logarithmic sweep, the number steps in one decade is 10, 25, or 50.
<b><i>Start</i></b>	Start value of sweep. Allowable range of <i>start</i> depends on output range of sweep source. For logarithmic sweep, <i>start</i> must not be zero.
<b><i>Stop</i></b>	Stop value of single sweep or turning back value of double sweep. Allowable range of <i>stop</i> depends on output range of sweep source. For logarithmic sweep, <i>stop</i> must have same polarity as <i>start</i> , and must not be zero.
<b><i>Step, No of Step</i></b>	<p>For linear sweep, <i>step</i> is step increment of sweep. <i>No of Step</i> is calculated from <i>start</i>, <i>stop</i>, and <i>step</i>.</p> <p>For logarithmic sweep, <i>step</i> is invalid. Number of sweep steps is calculated from <i>start</i>, <i>stop</i>, and number of steps in one decade.</p> <p>Calculated number of steps must be in range: 1 to 10001.</p>
<b><i>Compliance</i></b>	Compliance value of sweep source. Allowable range of <i>compliance</i> depends on the compliance range of sweep source.
<b><i>Pwr Comp</i></b>	(Optional) Power compliance value of sweep source. Allowable range depends on power compliance range of sweep source.
<b><i>Hold</i></b>	Time required for DUT to settle after forcing start value. 0 to 655.35 s with 10 ms resolution.
<b><i>Delay</i></b>	Time required for DUT to settle after stepping the output. 0 to 65.535 s with 100 $\mu$ s resolution.

Figure 4-3

### Basic Sweep Measurement



### Subordinate Sweep Measurement

For subordinate sweep measurement, you set up a secondary sweep source (VAR2) in addition to a primary sweep source (VAR1). After VAR1 sweep is completed, the output of VAR2 sweep source is increased or decreased by the specified step value, then the VAR1 sweep source is swept again.

Only the linear single sweep is available for the VAR2.

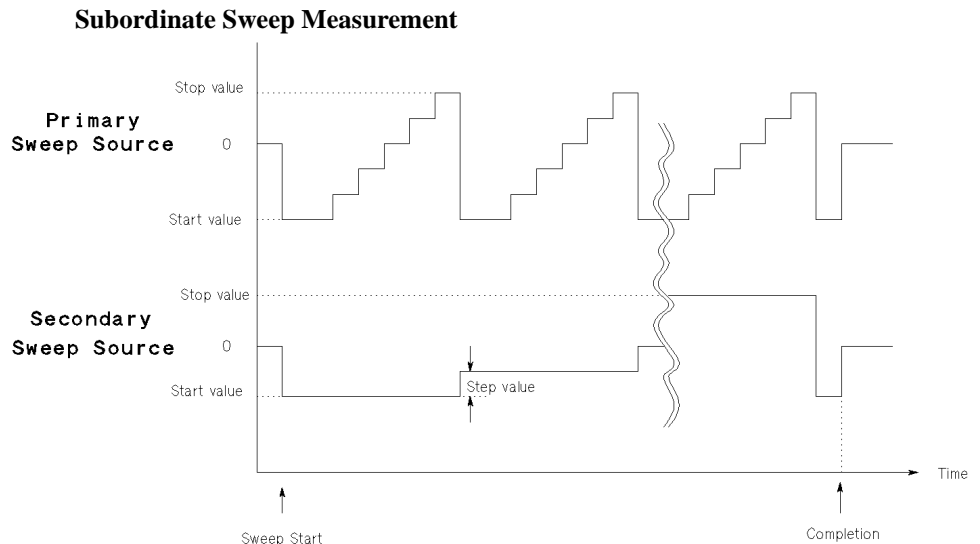
The following setup is required to perform the subordinate sweep measurements. See “I/V Sweep” on page 2-4 for the GUI.

1. Select the following on the Channel Setup tab screen.
  - Select VAR1 for the Function of the SMU used for VAR1.
  - Select VAR2 for the Function of the SMU used for VAR2.
  - Select V, VPULSE, or COMMON for the Mode of current monitor SMU, or I or IPULSE for voltage monitor SMU. Pulse is available only for one SMU.

2. Set the VAR1 channel as shown in “Basic Sweep Measurement”.
3. Specify the following parameters for VAR2 on the Measurement Setup tab screen.

<b>Start</b>	Start value of secondary sweep. Allowable range of <i>start</i> depends on the output range of secondary sweep source.
<b>Stop</b>	Stop value of secondary sweep is calculated from <i>start</i> , <i>step</i> , and <i>number of steps</i> . Allowable range of <i>stop</i> depends on the output range of secondary sweep source.
<b>Step</b>	Step increment of secondary sweep.
<b>No of Step</b>	Number of secondary sweep steps ( $N_2$ ). 1 to 1001. Then, the following formula must be satisfied. $1 \leq N_1 \times N_2 \leq 128128$ ( $N_1$ : number of primary sweep steps)
<b>Compliance</b>	Compliance value of secondary sweep source. Allowable range of <i>compliance</i> depends on the compliance range of secondary sweep source.
<b>Pwr Comp</b>	(Optional) Power compliance value of secondary sweep source. Allowable range of <i>power compliance</i> depends on the power compliance range of sweep source.

Figure 4-4



## Synchronous Sweep Measurement

For synchronous sweep measurement, you set up a synchronous sweep source (VAR1') in addition to a primary sweep source (VAR1).

The relationship between the output of primary and synchronous sweep sources is determined by the following equation:

$$\text{synchronous output} = \text{primary output} \times \text{ratio} + \text{offset}$$

The synchronous output determined by above equation must not exceed the output range of synchronous sweep source.

The following parameter settings are effective for the VAR1' sweep source.

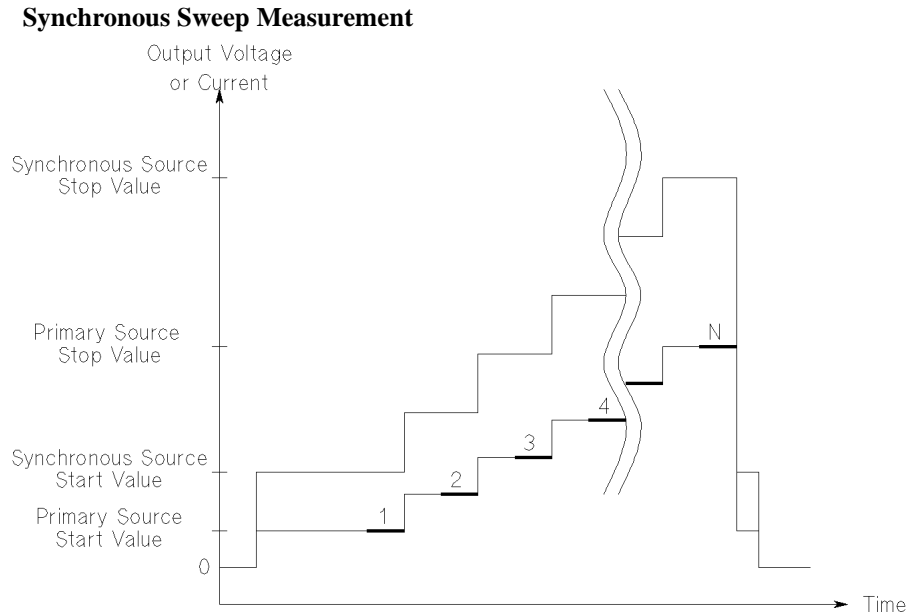
- Direction (Single or Double)
- Linear/Log

The following setup is required to perform the synchronous sweep measurements. See "I/V Sweep" on page 2-4 for the GUI.

1. Select the following on the Channel Setup tab screen.
  - Select VAR1 for the Function of the SMU used for VAR1.
  - Select VAR1' for the Function of the SMU used for VAR1'.
  - Select V, VPULSE, or COMMON for the Mode of current monitor SMU, or I or IPULSE for voltage monitor SMU. Pulse is available only for one SMU.Mode of VAR1 and VAR1' must be the same, voltage or current.
2. Set the VAR1 channel as shown in "Basic Sweep Measurement".
3. Specify the following parameters for VAR1' on the Measurement Setup tab screen.

<b>Offset</b>	Offset between outputs of primary and synchronous sweep sources.
<b>Ratio</b>	Ratio between outputs of primary and synchronous sweep sources.
<b>Compliance</b>	Compliance value of synchronous sweep source. Allowable range of <i>compliance</i> depends on the compliance range of synchronous sweep source.
<b>Pwr Comp</b>	(Optional) Power compliance value of synchronous sweep source. Allowable range of <i>power compliance</i> depends on the power compliance range of synchronous sweep source.

Figure 4-5



## Pulsed Sweep Measurement

For a sweep measurement, a sweep or constant source SMU can be a pulse source. But *only one* SMU can be a pulse source.

Figure 4-6 shows the relationship between pulse source and other sources. Each step of the primary sweep source is synchronized with output of the SMU pulse source. Measurements are made during the pulse output. For the pulsed sweep measurement, the delay time cannot be set.

The following setup is required to perform the pulsed sweep measurements. See “I/V Sweep” on page 2-4 for the GUI.

1. Select the following on the Channel Setup tab screen.
  - Select VAR1 for the Function of the SMU used for VAR1.
  - Select V, VPULSE, or COMMON for the Mode of current monitor SMU, or I or IPULSE for voltage monitor SMU. Pulse is available only for one SMU.
2. Set the VAR1 channel as shown in “Basic Sweep Measurement”.
3. Specify the following parameters for the SMU Pulse on the Measurement Setup tab screen.

Function Details  
I/V Sweep Measurement

<b>Period</b>	SMU forces the next pulse after specified <i>pulse period</i> .
<b>Width</b>	Time from when SMU output starts to change from <i>base value</i> to time when SMU starts to return from peak value. Measurements are made while the peak value is output.
<b>Base</b>	The base output value of the SMU pulse.

**Pulse peak value** Pulse peak value depends on the Function setting of the pulse source SMU. See Table 4-1.

**Table 4-1 Pulse Peak and Function**

<b>Function</b>	<b>Pulse peak</b>
CONST	Source value in the Constants area
VAR1	Calculated from the Start, Stop, Step values of the VAR1.
VAR2	Calculated from the Start, Stop, No of Step values of the VAR2.
VAR1 '	Calculated from the Start, Stop, Step values of the VAR1 and the Offset and Ratio values of the VAR1'.

**Measurement channel** Only one SMU can be used for the measurement channel.

The measurement channel always uses the compliance range which covers the setting compliance value. The Range button in the Measurement Setup is not available.

Figure 4-6

**Pulse Source and Other Sources**

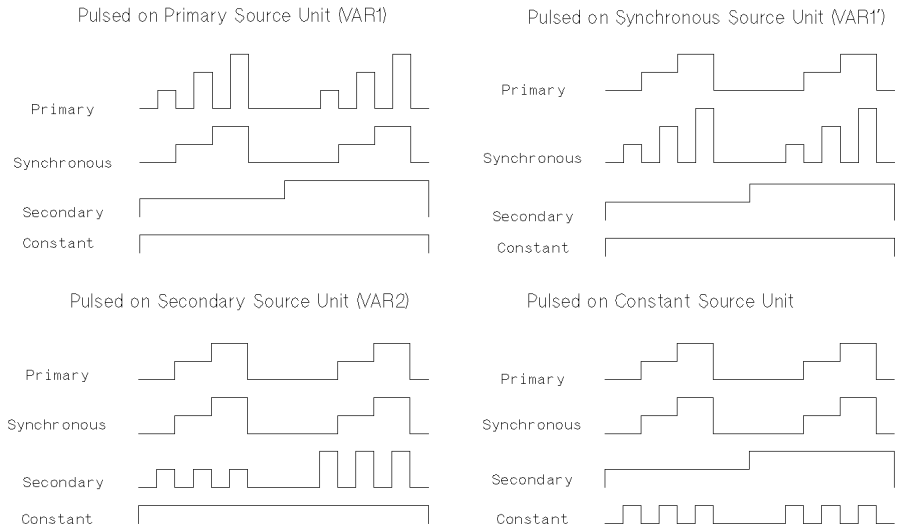
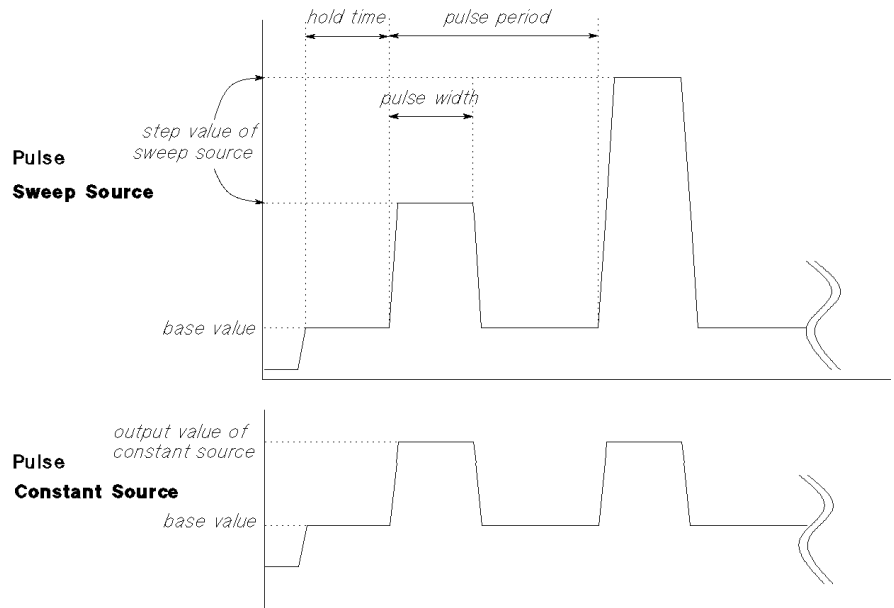


Figure 4-7

**SMU Pulse**



## Multi Channel I/V Sweep Measurement

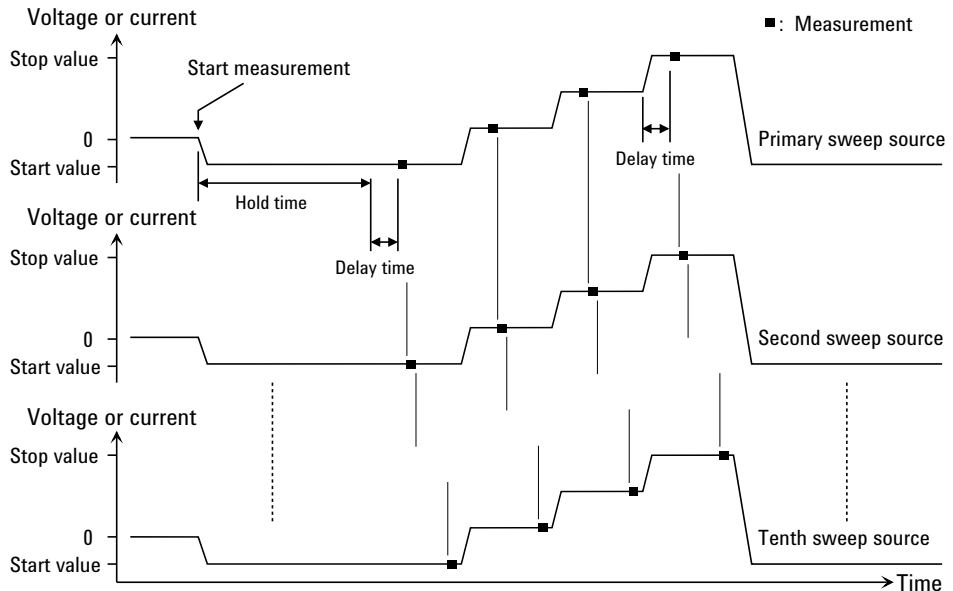
Multi Channel I/V Sweep measurement is similar to I/V Sweep measurement. Changes from I/V Sweep are listed below.

- All SMU can be set to the VAR1 sweep source (up to 10 channels)
- All SMU can be set to the pulse source (up to 10 channels)
- There is no restriction for the VAR1 output mode, voltage or current
- VAR1' sweep mode is not available

The timing of the sweep output and measurements are shown in Figure 4-8. The DC output channels start to output in the order in which they are listed on the Channel Setup screen. If multiple measurement channels are used, the channels perform the measurements in order. See “Multiple Measurement Channels” on page 4-69.

If multiple pulse output channels are used, the DC output channels start to output in the order in which they are listed on the Channel Setup screen, and then the pulse output channels start to output simultaneously. And the all measurement channels perform measurements simultaneously.

**Figure 4-8 Multi Channel I/V Sweep Measurements using High-Resolution A/D Converter**





## I/V-t Sampling Measurement

For the I/V-t sampling measurements, source channels (SMU) force constant current or voltage, and monitor channels (SMU) measure current or voltage changes at a device under test (DUT) with a specified sampling interval.

### Available Measurement Resources

SMUs and SPGUs can be used. SMU can be constant voltage source or constant current source and perform voltage measurement or current measurement. SPGU can be pulse voltage source. See “SPGU Module” on page 4-27 for using the SPGU.

#### NOTE

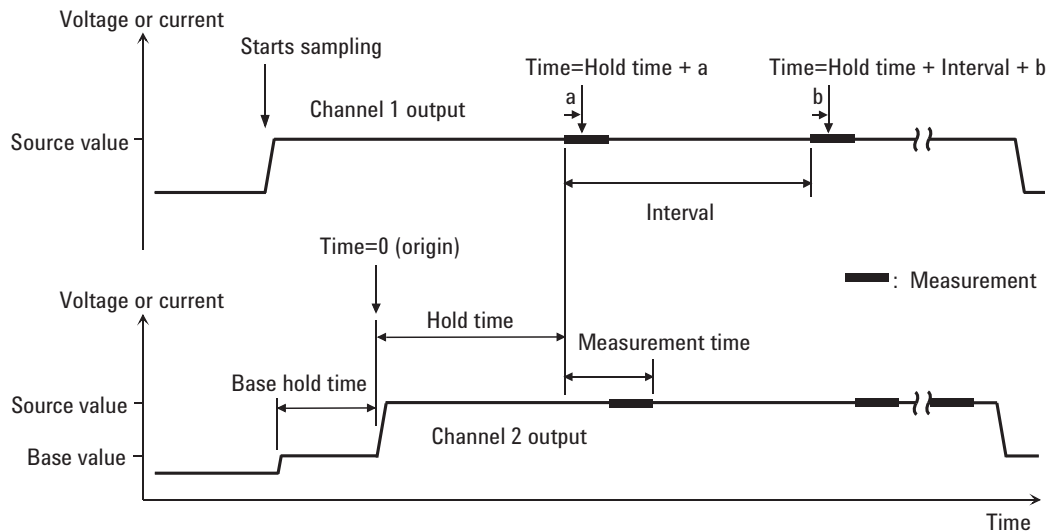
To use the SPGU, the sampling interval must be  $\geq 2$  ms.

### Operation Summary

I/V-t sampling measurement is executed as explained below. See Figure 4-9. In this example, the Output Sequence is set to SEQUENTIAL. And the bias hold function sets the bias to the *Base* value.

Figure 4-9

#### Operation Summary



In this figure, *a* and *b* in the formula of *Time* indicate the time from the start of one point measurement to the start of integration by the A/D converter.

## Function Details

### I/V-t Sampling Measurement

1. When the Output Sequence is SEQUENTIAL, source channels start the *Source* value output in order, and the last source channel starts the *Base* value output (Starts sampling). See “Source Output Sequence and Time Origin” on page 4-18.

When the Output Sequence is SIMULTANEOUS, source channels start the *Base* value output simultaneously (Starts sampling).

*Base* value is available for all output channels for the SIMULTANEOUS mode and the last source channel for the SEQUENTIAL mode.

2. *Base Hold Time* later, the source channels change the output to the *Source* value. The channels keep the value until the end of the sampling measurement.
3. Another *Hold Time* later, the measurement channel starts measurement for the first sampling point. If multiple channels are used, the channels perform measurement in order. See “Multiple Measurement Channels” on page 4-69.
4. After that, the following operation is repeated with the specified time *Interval*.

This operation is repeated until the number of measurement result data reaches the specified *No of Samples* of measurement data.

- Measurement channels start measurement if they are ready to measure.
- Measurement channels keep the condition if they are busy.

For the log sampling, the B1500 holds only the data that can be plotted on the log scale in the same distance as close as possible. Only the held data is counted in the number of measurement result data.

For the linear sampling with  $Interval < 2$  ms, if the total measurement time runs over the specified time  $Interval \times No\ of\ Samples$ , the sampling measurement will be stopped even if the number of measurement result data is less than the specified *No of Samples*.

5. The source channels stop the output, and the I/V-t sampling measurement is completed.

If the bias hold function is enabled, the source channels force the *Base* or *Source* value.

---

#### NOTE

#### Time data and index

For the log sampling, the recorded data will be only the data that can be plotted on the log scale in the same distance as close as possible. The index of the first data is 1. And the index of the following data will increase by 1. For the linear sampling, see “Time Data and Index” on page 4-17.

## Setup Parameters

The following setup is required to perform the I/V-t sampling measurements. See “I/V-t Sampling” on page 2-21 for the GUI.

---

**NOTE**

If you set  $Interval < 0.002$  s

Sampling mode must be linear. This setting is not permitted for the log sampling. Also SPGU is not available.

All measurement channels must use the high speed A/D converter (ADC). This setting is not permitted if a measurement channel uses the high resolution ADC.

If the multiple measurement channels are used, all channels perform measurement in parallel.

If the measurement ranging mode is not the fixed mode, the measurement channels automatically select the minimum range that covers compliance value set to the channel.

If the measurement time is expected to be longer than *Interval*, the measurement channels automatically adjust the number of averaging samples (ADC settings) to keep the sampling interval.

- 
1. Select the following on the Channel Setup screen.
    - Select SMUs used for constant voltage sources or constant current sources.
    - Select V or COMMON for Mode of current monitor SMU, or I for voltage monitor SMU.
  2. Specify the following parameters for the SMU in the Sampling Parameter area on the Measurement Setup screen.

***Linear/Log***

Sampling mode. LINEAR for linear sampling. LOG10, LOG25, LOG50, LOG100, LOG250, or LOG500 for logarithmic sampling.

For the logarithmic sampling, the number after LOG indicates the number of measurement data in a decade. For example, LOG10 mode is used to get 10 data/decade.

***Interval***

Interval of the sampling, in seconds. 0.002 to 65.535 s, 0.001 s resolution.

$Interval < 0.002$  s in 0.00001 s resolution is also available for the linear sampling. It must satisfy the following formula. See NOTE above.

## Function Details

### I/V-t Sampling Measurement

$Interval \geq 0.0001 + 0.00002 \times (\text{number of measurement channels} - 1)$

**No of Samples** Number of samples. Integer. 1 to the following value.  
For linear:  $100001 / (\text{number of measurement channels})$   
For logarithmic:  $1 + (\text{number of data for 11 decades})$

**Total Sampling Time** This field just displays the total sampling time that is the time from the measurement start time for the first point to the end of sampling measurement. It does not include the hold time.

$Total\ Sampling\ Time = Interval \times No\ of\ Samples$

**Output Sequence** Source output sequence. SIMULTANEOUS or SEQUENTIAL. See “Source Output Sequence and Time Origin” on page 4-18.

**Hold Time** Time since the *Source* value output until the first sampling point, in seconds. 0 to 655.35 s, resolution 0.01 s.  
The following values are also available for  $Interval < 0.002$  s.  $|Hold\ Time|$  will be the time since the sampling start until the *Source* value output.  
-0.09 to -0.0001 s, resolution 0.0001 s.

**Base Hold Time** Hold time of the *Base* value output until the *Source* value output, in seconds. 0 to 655.35 s, resolution 0.01 s.

- Specify the following parameters for the SMU in the Constants area on the Measurement Setup screen.

**Base, Source** Base or Source value, in V or A.  
*Base* value is available for all output channels for the SIMULTANEOUS mode and the last source channel for the SEQUENTIAL mode.

**Compliance** Compliance value, in A or V.

## Time Data and Index

To obtain the time data, enter a variable name (e.g. Time) into the Time Stamp Name field of the Channel Setup screen. The time data can be obtained after the *linear* sampling measurement is executed, and can be expressed by the following formula.

$$\text{Time} = \text{Hold Time} + \text{Interval} \times [(\text{Index} - 1) + N] + \alpha$$

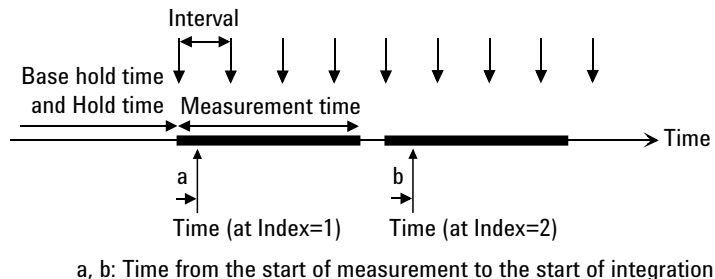
- Time :** Variable name set to the Time Stamp Name field of the Measurement Setup screen. Time data is returned.
- Index :** Variable name set to the Index Name field of the Measurement Setup screen. Data index is returned. Maximum value is 2147483647.
- Interval :** Sampling interval. Defined in the Measurement Setup screen.
- Hold Time :** Hold time. Defined in the Measurement Setup screen.
- N :** Number of samplings performed while one point measurement was executed. This value is 0 if the sampling interval is longer than the measurement time.
- $\alpha$  :** Time from the start of one point measurement to the start of integration by the A/D converter.

If the sampling interval is longer than the actual measurement time, measurement channels repeat measurement with the specified sampling interval. However, if the sampling interval is shorter than the measurement time, the measurement interval will be integral multiples of the specified sampling interval. Example for *Hold Time* = 10 ms, *Interval* = 5 ms, and the following *Time* values is shown in Figure 4-10. In this case,  $N=3$ .

- Time (for Index=1) = 10 ms = 10 + 5 × [(1 - 1) + 0] ms
- Time (for Index=2) = 30 ms = 10 + 5 × [(2 - 1) + 3] ms

Figure 4-10

### Sampling and Measurement Timing



## Source Output Sequence and Time Origin

When the Output Sequence is SEQUENTIAL, source channels start output in order. The order is top to bottom of the channels defined in the Channel Setup screen. The order of bias stop is bottom to top.

When the Output Sequence is SIMULTANEOUS, source channels start output simultaneously. And the source channels stop output at the same time.

The time origin is when the source output value is changed from *Base* to *Source*.

## Sampling Completion

The sampling measurement completes when one of the following conditions is satisfied:

- The event of the stop condition occurs. See “Stop Condition” on page 4-18.
- The measurement channels obtain the specified number (*No of Samples*) of measurement data.
- The total measurement time runs over the time  $Interval \times No\ of\ Samples$  for the linear sampling with  $Interval < 2\ ms$ .
- The Stop button is clicked.
- An emergency condition occurs on the B1500.
- Interlock terminal is opened when a channel is in the high voltage output status.

## Stop Condition

The stop condition is defined by using the Stop Condition fields on the Measurement Setup screen. This function stops the measurement as shown below.

1. Compares the value of parameter set to the Name field and the value defined in the Threshold field.
2. Counts how many times the selected Event occurs.
3. When the count reaches the value defined in the No. of Events field, sampling is stopped immediately.

For the setup parameters of the stop condition, see “I/V-t Sampling” on page 2-21.

## C-V Sweep Measurement

For the C-V sweep measurements, the multi frequency capacitance measurement unit (MFCMU) forces AC signal, performs staircase sweep output of DC bias voltage, and measures impedance for each sweep step.

### Available Measurement Resources

MFCMU can be used for the C-V sweep measurements. For the DC bias output over  $\pm 25$  V, use the SMU CMU Unify Unit (SCUU) and SMUs. This expands the voltage range up to  $\pm 100$  V. Or use the high voltage bias-T and HVSMU to expand the voltage range up to  $\pm 3000$  V.

### Setup Parameters

The following setup is required to perform the C-V sweep measurements. See “C-V Sweep” on page 2-26 for the GUI.

1. Set the following on the Channel Setup tab screen.
  - Set V Name of the CMU.
  - If you use a constant current or voltage source, set Mode (I, V, or COMMON) and V Name or I Name of the SMU.
2. Specify the following parameters for the CMU on the Measurement Setup tab screen.

<b><i>Model, x Name</i></b>	Measurement parameters. See Table 4-2.
<b><i>Direction</i></b>	Single or double sweep.
<b><i>Linear/Log</i></b>	Linear or logarithmic sweep. For logarithmic sweep, the number steps in one decade is 10, 25, or 50.
<b><i>Start, Stop</i></b>	<p><i>Start</i> is the DC bias sweep start value. <i>Stop</i> is the stop value of single sweep or turning back value of double sweep. The CMU can force up to <math>\pm 25</math> V.</p> <p>If the SCUU (SMU CMU Unify Unit) is used, up to <math>\pm 100</math> V output is available.</p> <p>If the high voltage bias-T and HVSMU are used, up to <math>\pm 3000</math> V output is available.</p>

## Function Details

### C-V Sweep Measurement

#### ***Step, No of Step***

*Step* is step increment of sweep. *No of Step* value is calculated from *start*, *stop*, and *step*. Calculated number of steps must be in range: 2 to 1001.

#### ***Frequency List***

AC signal frequency. 1 kHz to 5 MHz. Resolution: 6 digits.

#### ***F Name***

Variable name of frequency list.

#### ***AC Level***

AC signal level. 10 mV to 250 mV, 1 mV resolution.

#### ***Mode, Factor***

One of the following is used for setting the measurement time.

For *Mode*=AUTO, you set the number of samples used for averaging of the measurement data. The initial value is the predefined constant, and you set the multiple number to *Factor*. Available *Factor* values are 1 to 1023, integer.

For *Mode*=PLC, you set the integration time of the A/D converter. The initial value is 1 power line cycle (20 ms for 50 Hz, 16.7 ms for 60 Hz), and you set the multiple number to *Factor*. Available *Factor* values are 1 to 100, integer.

#### ***Hold***

Hold time. The time required for DUT to settle after forcing start value. 0 to 655.35 s with 10 ms resolution.

#### ***Delay***

Time required for DUT to settle after stepping the DC bias output value. 0 to 65.535 s with 100  $\mu$ s resolution.

---

#### **NOTE**

For a more accurate measurement, perform correction data measurement at the measurement frequency before starting the capacitance measurement.

If the measurement frequency is not included in the list of default frequencies below, click the Advanced Options... button and set the measurement frequency on the Frequency area of the Advanced Options for CMU Calibration window.

Default frequencies:

1 k, 2 k, 5 k, 10 k, 20 k, 50 k, 100 k, 200 k, 500 k, 1 M, 1.2 M, 1.5 M, 2 M, 2.5 M, 2.7 M, 3 M, 3.2 M, 3.5 M, 3.7 M, 4 M, 4.2 M, 4.5 M, 5 MHz

---



**Table 4-2 MFCMU Measurement Parameters**

Primary Parameter	Secondary Parameter
R (resistance, $\Omega$ )	X (reactance, $\Omega$ )
G (conductance, S)	B (susceptance, S)
Z (impedance, $\Omega$ )	$\theta$ (phase, radian)
Z (impedance, $\Omega$ )	$\theta$ (phase, degree)
Y (admittance, S)	$\theta$ (phase, radian)
Y (admittance, S)	$\theta$ (phase, degree)
Cp (parallel capacitance, F)	G (conductance, S)
Cp (parallel capacitance, F)	D (dissipation factor)
Cp (parallel capacitance, F)	Q (quality factor)
Cp (parallel capacitance, F)	Rp (parallel resistance, $\Omega$ )
Cs (series capacitance, F)	Rs (series resistance, $\Omega$ )
Cs (series capacitance, F)	D (dissipation factor)
Cs (series capacitance, F)	Q (quality factor)
Lp (parallel inductance, H)	G (conductance, S)
Lp (parallel inductance, H)	D (dissipation factor)
Lp (parallel inductance, H)	Q (quality factor)
Lp (parallel inductance, H)	Rp (parallel resistance, $\Omega$ )
Ls (series inductance, H)	Rs (series resistance, $\Omega$ )
Ls (series inductance, H)	D (dissipation factor)
Ls (series inductance, H)	Q (quality factor)

## Measurement Parameters

MFCMU performs impedance measurement, calculates parameters shown in Table 4-2, and returns the calculated data. A combination can be selected for the return data. For example and for your reference, select the parallel measurement mode (Cp-G or Cp-D) for the low capacitance measurements (100 Ω or more of impedance), and select the series measurement mode (Cs-Rs) for the high capacitance measurements (100 Ω or less of impedance). Because the low capacitance yields high reactance, which implies that the effect of the parallel resistance has relatively more significance than that of series resistance.

## About Impedance Measurements

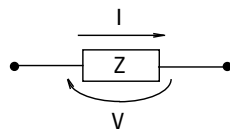
All circuit components, resistors, capacitors, or inductors, have parasitic components, for example unwanted resistance in capacitors, unwanted resistance in inductors, and unwanted inductance in resistors. Thus simple components should be modeled as complex impedances.

Impedance definitions and vector representation of impedance are shown in Figure 4-11. Impedance  $Z$  is the total opposition that a circuit or device offers to the flow of alternating current at a given frequency.  $Z$  contains a real and an imaginary part, and it is expressed in rectangular form as resistance  $R$  and reactance  $X$ , or in polar form as magnitude of impedance  $|Z|$  and phase  $\theta$ .

In addition to these parameters, the quality factor  $Q$  and dissipation factor  $D$  are used to describe the quality of components.

Figure 4-11

### Impedance and Parameter Calculation



$$Z = \frac{V}{I}$$

( I: current (A)  
V: voltage (V)

$$Z = R + jX = |Z| \angle \theta$$

$$|Z| = \sqrt{R^2 + X^2}$$

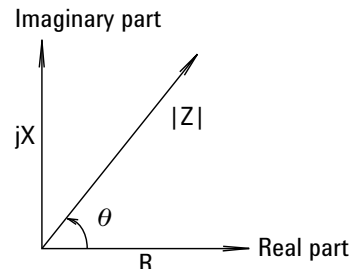
$$\theta = \tan^{-1}\left(\frac{|X|}{R}\right)$$

$$R = R_s$$

$$X = 2\pi f L$$

$$Q = \frac{1}{D} = \frac{|X|}{R}$$

- Z: impedance (Ω)
- R: resistance (Ω)
- X: reactance (Ω)
- θ: phase (degree or radian)
- Rs: series resistance (Ω)
- f: frequency (Hz)
- L: inductance (H)
- Q: quality factor
- D: dissipation factor



For your reference, select the parallel measurement mode (Lp-G, Lp-D, or anything) for the high inductance measurements (100 Ω or more of impedance), and select the series measurement mode (Ls-Rs, or anything) for the low inductance measurements (100 Ω or less of impedance). Because the reactance of high inductance at a given frequency is relatively high (compared with that of a low inductance), so the parallel resistance becomes more significant than the series component.

Figure 4-12

### Admittance and Parameter Calculation

$$Y = \frac{1}{Z} = G + jB = |Y| \angle \phi$$

$$|Y| = \sqrt{G^2 + B^2}$$

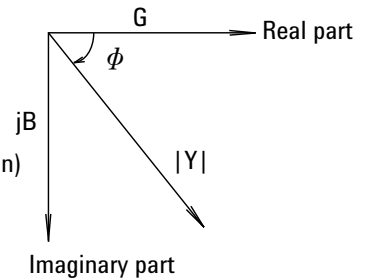
$$\phi = \tan^{-1} \left( \frac{|B|}{G} \right) = -\theta$$

$$B = 2\pi f C$$

$$Q = \frac{1}{D} = \frac{|B|}{G}$$

$$G = \frac{1}{R_p}$$

- Y: admittance (S)
- G: conductance (S)
- B: susceptance (S)
- $\phi$ : phase (degree or radian)
- f: frequency (Hz)
- C: capacitance (F)
- Q: quality factor
- D: dissipation factor
- Rp: parallel resistance (Ω)



In some case, the reciprocal of impedance (admittance) Y is used. Figure 4-12 shows the vector representation of admittance. As impedance Z, Y contains a real and an imaginary part, and is expressed in rectangular form as conductance G and susceptance B, or in polar form as magnitude of admittance |Y| and phase  $\phi$ .

## Four-Terminal Pair Configuration

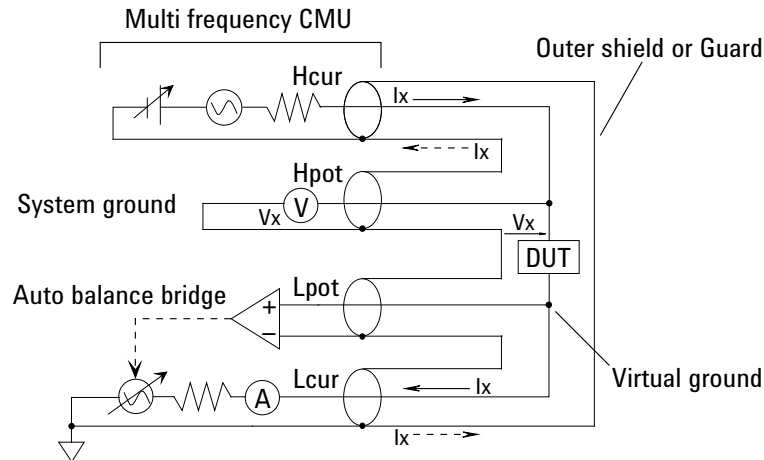
Generally, any mutual inductance, interference of the measurement signals, and unwanted residual factors in the connection method incidental to ordinary termination methods will have significant effects on the measurements, especially at a high frequency. The MFCMU employs the four-terminal pair (4TP) measurement configuration which permits easy, stable, and accurate measurements and avoids the measurement limitations inherent to such factors.

Figure 4-13 shows the four-terminal pair measurement principle. The measurement terminals consist of the following four coaxial connectors.

- Hcur: High current
- Hpot: High potential
- Lpot: Low potential
- Lcur: Low current

Figure 4-13

### Four-Terminal Pair Measurement Principle



The four-terminal pair measurement method has the advantage in both low and high impedance measurements. The outer shield conductors work as the return path for the measurement signal current (they are not grounded). The same current flows through both the center conductors and outer shield conductors (in opposite directions), but no external magnetic fields are generated around the conductors (the magnetic fields produced by the inner and outer current completely cancel each other). Because the measurement signal current does not develop an inductive magnetic field, test leads do not contribute additional errors due to self or mutual inductance between the individual leads.

To realize accurate measurements using the four-terminal pair measurement method, the following connection tips are important. See Figure 4-13 and Figure 4-14.

- The signal path from MFCMU to DUT must be as short as possible.
- The four-terminal pair configuration must be extended as close as possible to the DUT.
- The outer shield conductors must be connected together at the extended cable ends. Then take care of contacts. They must be electrically isolated.

This is the same as connecting the CMU return terminals of the atto sense/switch unit (ASU) or the guard switch unit (GSWU).

- The signal path from the shield ends to the DUT must be as short as possible.
- If it is possible, make protective guard (plate, shield, or anything) around the DUT, and connect it to the outer shield conductor of the four-terminal pair configuration. This minimizes the effects of stray capacitance and electrical noise. Then take care of contacts. The guard must be electrically isolated.
- The Low (Lcur, Lpot) terminal must be prevented from stray capacitance and guard capacitance. If the measurement terminal has to be connected to the wafer chuck, use the High (Hcur, Hpot) terminal to connect to the wafer chuck.

## Error Correction

The MFCMU is equipped with the error correction function used to realize accurate impedance measurements. The correction function minimizes the effects of the error elements in the extension cables and the DUT interface such as manipulator and probe card.

- Phase compensation  
Corrects phase error caused by extending measurement cables.
- Open correction  
Corrects for stray admittance. Effective for high impedance measurements.
- Short correction  
Corrects for residual impedance. Effective for low impedance measurements.
- Load correction  
Corrects any error by using the working standard.

## Function Details

### C-V Sweep Measurement

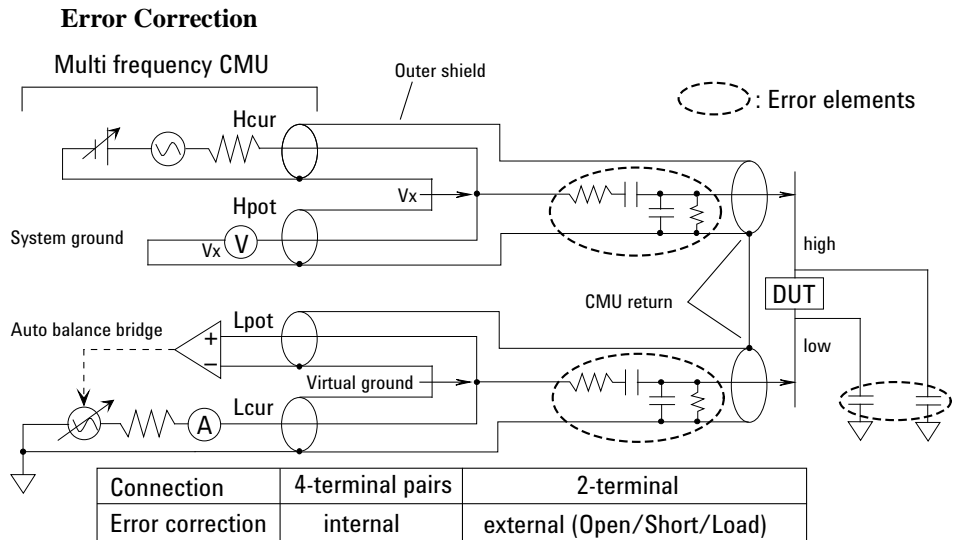
The MFCMU will internally perform the error correction for the CMU cables and CMU accessories that keep the four-terminal pair configuration.

For the measurements for evaluating DUT against a working standard, perform the open/short/load correction.

However, you have to consider that some parasitic remain in measurement path even after performing corrections. To minimize the influence of parasitic, make the signal path from the shield ends to the DUT as short as possible.

To perform the phase compensation and error correction, see “CMU Calibration” on page 1-60.

**Figure 4-14**



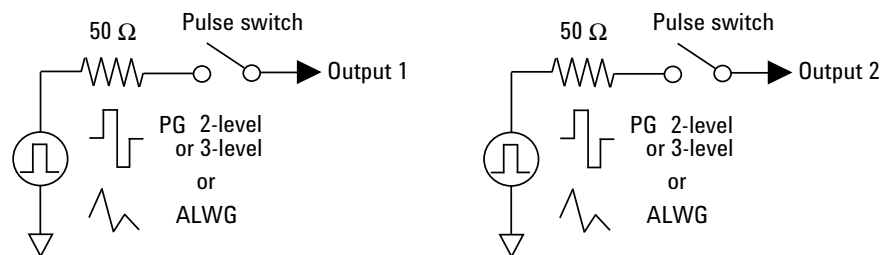
## SPGU Module

Only for the B1500A. High voltage SPGU is a pulse generator module designed for semiconductor parametric test applications and provides the following key functions. The SPGU can be controlled by using the SPGU Control classic test or the SPGU setup function in the I/V Sweep, Multi Channel I/V Sweep, I/V List Sweep, and I/V-t Sampling classic tests.

- Number of output channels: 2 channels per module
- Output impedance:  $50\ \Omega$
- Output level: 0 to  $\pm 40\ \text{V}$  (open load),  $\pm 20\ \text{V}$  ( $50\ \Omega$  load)
- SPGU operation mode (effective for all channels installed in B1500A):  
PG (pulse generator) or ALWG (arbitrary linear waveform generator)
- Channel output operation mode:  
Free run, duration, or pulse count (PG mode)/sequence count (ALWG mode)  
Free run mode is not available for the SPGU Control classic test.
- PG output mode:  
2-level pulse or 3-level pulse, for each channel
- Pulse switch, which is more durable than mechanical relays, and better suited for frequent switching applications.
- Automatic voltage level adjustment by specifying load impedance of DUT

Figure 4-15

### SPGU Simplified Circuit Diagram



#### NOTE

To use the SPGU for I/V-t Sampling tests, the sampling interval must be  $\geq 2\ \text{ms}$ .

## PG Operation Mode

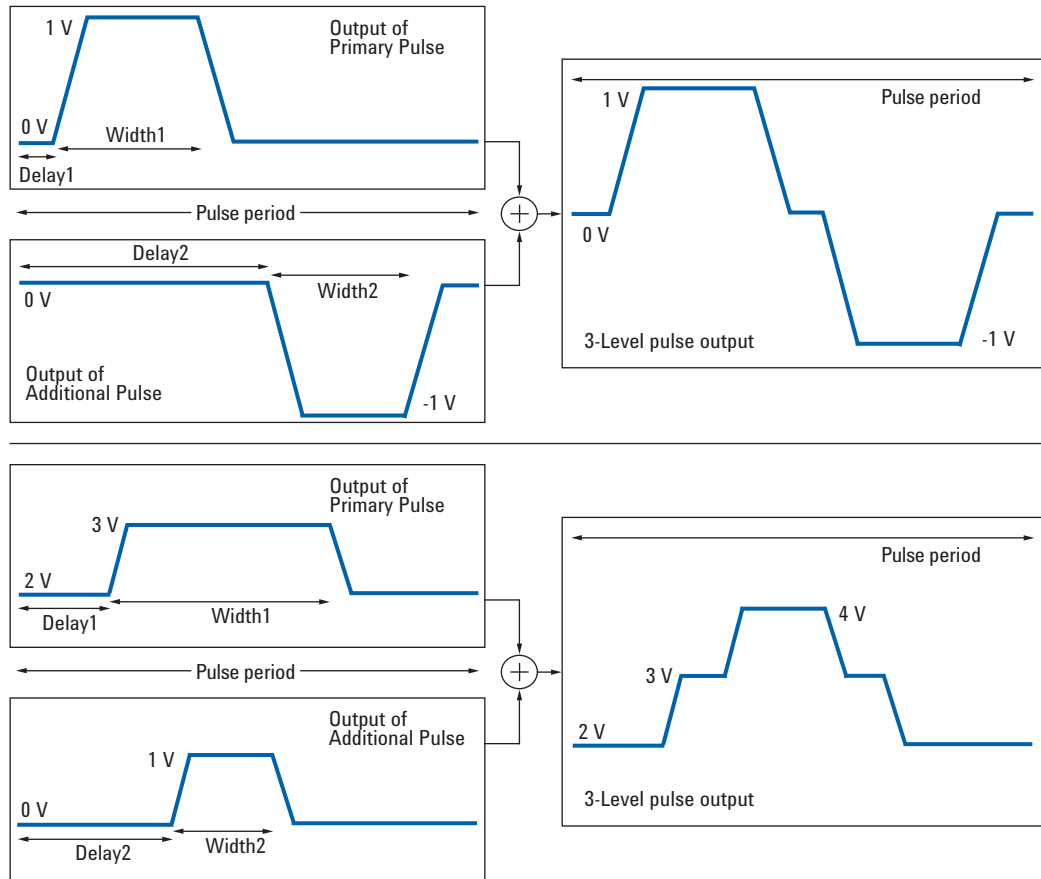
In the PG mode (pulse generator operation mode), the SPGU outputs 2-level or 3-level pulse voltage as shown in Figure 4-16.

The SPGU channels simultaneously start the pulse output and repeat the output every pulse period. The channels continue the pulse output until the specified time elapses or the repeat count reaches the specified number of pulses.

For the SPGU pulse setup, see “SPGU Pulse Setup Parameters” on page 4-29.

For a summary of source output operation in sweep or sampling measurement mode, see “Pulse Output Operation Summary” on page 4-31.

**Figure 4-16** 3-Level Pulse Output Examples





## SPGU Pulse Setup Parameters

The following setup is required to apply the SPGU pulse. See “SPGU Control” on page 2-61 for the GUI.

1. Set the following for the SPGU channel on the Channel Setup tab screen.
  - Set the V Name of the SPGU.
  - Set the Mode to VPULSE. This sets the PG mode. Do not set to ALWG.
2. On the SPGU Pulse Setup window, specify the following parameters for the SPGU. See Figure 4-17 for the pulse setup parameters.

<b><i>Period</i></b>	Pulse period. 20 ns to 10 s, 10 ns resolution.
<b><i>Operation</i></b>	FREE RUN, PULSE COUNT, or DURATION.  In the FREE RUN mode, the SPGU continues the pulse output until the measurement is completed or stopped. Not available for the SPGU Control classic test.  For the PULSE COUNT mode, specify the number of pulses, within the range of 1 to 1000000. The SPGU will output the specified number of pulses.  For the DURATION mode, specify the total time to output, within the range of 1 $\mu$ s to 31556926 s (1 year). The SPGU will continue to output until the specified time elapses.
<b><i>Type</i></b>	2-Level Pulse or 3-Level Pulse. If you choose 3-Level Pulse, the Additional Pulse fields become effective.
<b><i>Base, Peak</i></b>	Pulse base and peak voltage. 0 to $\pm$ 40 V (open load), $\pm$ 20 V (50 $\Omega$ load), 1 mV resolution.
<b><i>Delay</i></b>	Delay time. 0 s to <i>Period</i> -20 ns, 2.5 ns resolution. Resolution will be set at 10 ns if the pulse transition time is $>$ 8 $\mu$ s.
<b><i>Width</i></b>	Pulse width. 10 ns to <i>Period</i> -10 ns, 2.5 ns resolution. Resolution will be set at 10 ns if the pulse transition time is $>$ 8 $\mu$ s.
<b><i>Leading, Trailing</i></b>	Pulse transition time. 8 ns to 400 ms, 2 ns resolution. Resolution will be set at 8 ns if the pulse transition time is $>$ 8 $\mu$ s.

## Function Details

### SPGU Module

- (Optional) Set the Additional Pulse fields for the 3-Level Pulse. 3-level pulses can be achieved by the Primary Pulse and the Additional Pulse. See Figure 4-16 for examples.
- On the Load Z Setup window, specify the load impedance of the DUT (device under test). The value is used to adjust the SPGU output voltage automatically. The SPGU will output the voltage close to the specified *Base* or *Peak* value.
- (Optional) On the Pulse Switch Setup window, enable the pulse switch, which is the semiconductor relay mounted in the SPGU output and is used to open the SPGU channel output. This relay is more durable than mechanical relays, and is better suited for frequent switching applications.

**SW Sync**            ENABLE (enables the pulse switch) or DISABLE (disables the pulse switch).

Switching of the pulse switch status (open and close) will be controlled automatically and synchronized with the output pulse, as shown in Figure 4-17.

*Delay*, *Width*, and *Normal* fields will be active when *SW Sync*=ENABLE.

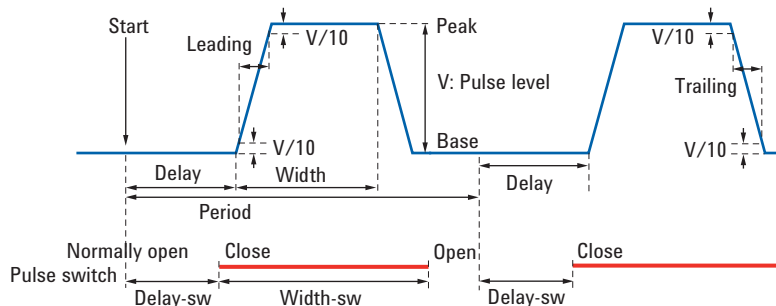
**Delay**                Delay time from start of pulse output to changeover of pulse switch. 0 s to *Period*-100 ns, 10 ns resolution.

**Width**              Duration to hold the switched state of the pulse switch. 100 ns to *Period*-*Delay* ns, 10 ns resolution.

**Normal**             Switch status in the pulse switch normal condition. CLOSE (normally closes the pulse switch) or OPEN (normally opens the pulse switch). The pulse switch will change the status after the *Delay-sw* time and keep the status until the *Width-sw* time elapses, as shown in Figure 4-17. This operation is repeated every pulse period.

Figure 4-17

### Pulse Setup Parameters



## Pulse Output Operation Summary

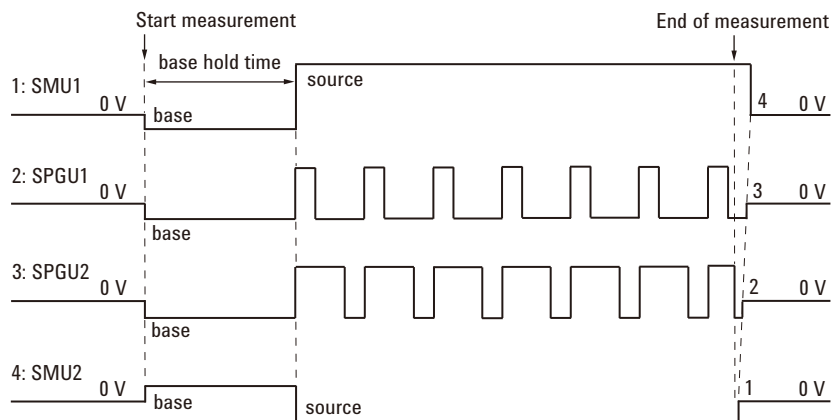
The SPGU can be used for I/V Sweep, Multi Channel I/V Sweep, I/V List Sweep, and I/V-t Sampling classic tests. The SPGU pulse output operation will differ as shown in the following cases. Pulse output example of the sampling measurement is shown in Figure 4-18.

- Sampling measurement, simultaneous output mode. See Table 4-3.
- Sampling measurement, sequential output mode. See Table 4-4
- Sweep measurement mode. See Table 4-5.

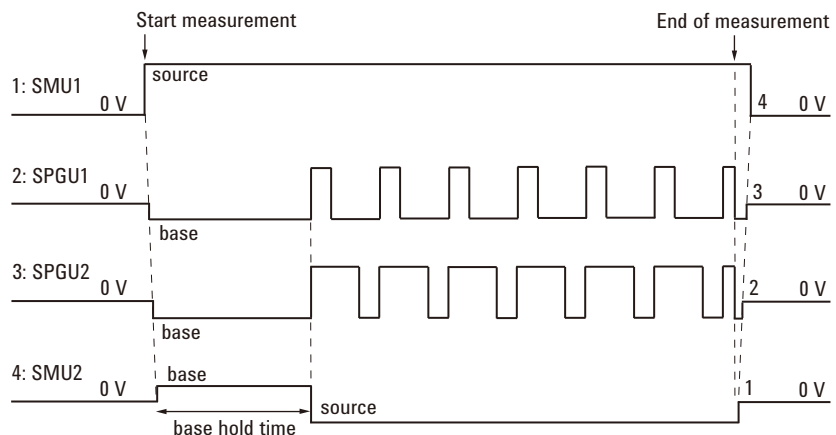
**Figure 4-18**

### SPGU Pulse Output Example for Sampling Measurement in Free Run Mode

Simultaneous output:



Sequential output:



**Table 4-3 Output Timing for I/V-t Sampling, Simultaneous**

Pulse output conditions	Source Channel Operation			
	Starting	During measurement	After measurement	Ending
Duration > Measurement time	SMUs and SPGUs start the base value output simultaneously .  SMUs start sampling measurement operation.	After the base hold time, SMUs start the source value output and SPGUs start pulse output simultaneously.	SMUs continue the source value output and SPGUs continue pulse output until the specified time elapses or the repeat count reaches the specified number.  After that, SMUs output the base or source value set by the Output Value after Measurement, and SPGUs output the base value.	If the bias hold function is ON, all source channels continue the output.  Else, all source output goes to 0 V in the reverse order of the Channel Setup definition.
Free run			SMUs output the base or source value set by the Output Value after Measurement, and SPGUs continue pulse output.  If the bias hold function is OFF, SPGUs output the base value.	
Duration < Measurement time		After the base hold time, SMUs start the source value output and SPGUs start pulse output simultaneously.  After the specified time elapses or the repeat count reaches the specified number, SPGUs output the base value.	SMUs output the base or source value set by the Output Value after Measurement, and SPGUs output the base value.	

**Table 4-4 Output Timing for I/V-t Sampling, Sequential**

Pulse output conditions	Source Channel Operation			
	Starting	During measurement	After measurement	Ending
Duration > Measurement time	Except for the last SMU, SMUs start the source value output. And SPGUs and the last SMU start the base value output. The source output sequence is the definition order on the Channel Setup screen (top to bottom).  SMUs start sampling measurement operation.	After the base hold time, the last SMU start the source value output and SPGUs start pulse output simultaneously.	SMUs continue the source value output and SPGUs continue pulse output until the specified time elapses or the repeat count reaches the specified number.  After that, the last SMU outputs the base or source value set by the Output Value after Measurement, and SPGUs output the base value.	If the bias hold function is ON, all source channels continue the output.  Else, all source output goes to 0 V in the reverse order of the Channel Setup definition.
Free run			The last SMU outputs the base or source value set by the Output Value after Measurement, and SPGUs continue pulse output.  If the bias hold function is OFF, SPGUs output the base value.	
Duration < Measurement time		After the base hold time, the last SMU start the source value output and SPGUs start pulse output simultaneously.  After the specified time elapses or the repeat count reaches the specified number, SPGUs output the base value.	The last SMU outputs the base or source value set by the Output Value after Measurement, and SPGUs output the base value.	

**Table 4-5 Output Timing for I/V Sweep, Multi Channel I/V Sweep, I/V List Sweep**

Pulse output conditions	Source Channel Operation			
	Starting	During measurement	After measurement	Ending
Duration > Measurement time	<p>SMUs and SPGUs start output. The source output sequence is the definition order on the Channel Setup screen (top to bottom).</p> <p>Then, sweep sources output the start value, constant sources output the source value, and</p>	<p>SPGUs start and continue the pulse output.</p> <p>SMUs start and perform sweep measurement operation.</p>	<p>Sweep sources output the start or stop value set by the Output Value after Measurement.</p> <p>SPGUs continue the pulse output until the specified time elapses or the repeat count reaches the specified number. After that, SPGUs output the base value.</p>	<p>If the bias hold function is ON, all source channels continue the output.</p> <p>Else, all source output goes to 0 V in the reverse order of the Channel Setup definition.</p>
Free run			<p>Sweep sources output the start or stop value set by the Output Value after Measurement.</p> <p>SPGUs continue pulse output.</p> <p>If the bias hold function is OFF, SPGUs output the base value.</p>	
Duration < Measurement time	<p>SPGUs output the base value.</p> <p>The constant sources continue the output until Ending.</p>	<p>SPGUs start the pulse output.</p> <p>SMUs start and perform sweep measurement operation.</p> <p>SPGUs continue the pulse output until the specified time elapses or the repeat count reaches the specified number. After that, SPGUs output the base value.</p>	<p>Sweep sources output the start or stop value set by the Output Value after Measurement.</p> <p>SPGUs output the base value.</p>	

## ALWG Operation Mode

In the ALWG mode (arbitrary linear waveform generator operation mode), SPGU can output an arbitrary linear waveform voltage as shown in Figure 4-19. The waveform is the voltage pattern sequence specified by the pattern data and the sequence data, which can be created by using the Define ALWG Waveform Window (p. 2-68).

For a summary of source output operation in sweep or sampling measurement mode, see “Pulse Output Operation Summary” on page 4-31. One sequence output will be handled as same as one pulse output.

### Pattern data

Specifies all of the voltage patterns set to a channel. A pattern consists of some vector data. A vector data is defined by the output level and the time data.

- Number of patterns: maximum 512 for a channel
- Number of vector data: maximum 1024-N (N: number of patterns) for defining all patterns to a channel
- Output level: 0 to  $\pm 40$  V (open load, 1 mV resolution)
- Time data (incremental time): 10 ns to 671.088630 ms (10 ns resolution)

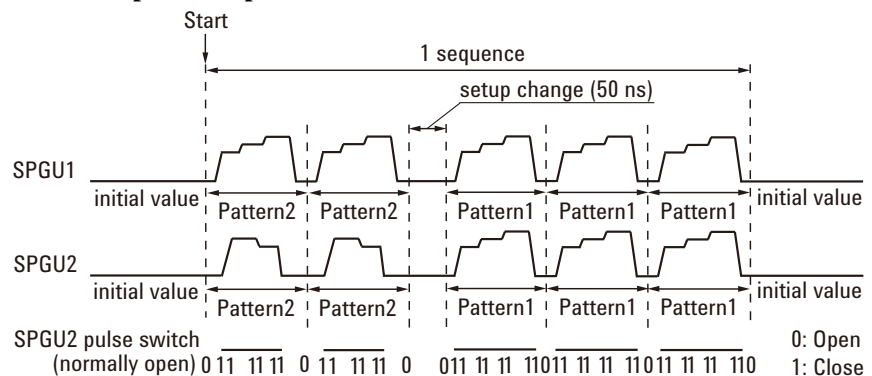
### Sequence data

Specifies the output channel, the output pattern, the repeat count of the pattern, and the output sequence of patterns applied in a single sequence.

- Repeat count of a pattern: 1,048,576
- Number of patterns in a sequence: maximum 512

Figure 4-19

### ALWG Output Example



## Function Details

### SPGU Module

In the example in Figure 4-19, SPGU1 and SPGU2 are used. The channels output Pattern2 twice and Pattern1 three times in one sequence, although the two patterns for SPGU1 are the same.

The SPGU channels simultaneously start the sequence output. The sequence will be repeated in the same way as for the pulse period. The channels continue the sequence output until the specified time elapses or the repeat count reaches the specified number of sequences.

---

#### NOTE

#### Setup delay time between the different patterns

When the pattern is changed to a pattern of a differing index, a 50 ns setup delay will always be inserted.

There is no delay between patterns of the same index.

---



---

## Sweep Abort Function

This function is used to stop sweep measurement automatically when one of the following conditions is detected. This function is useful to reduce sweep time and to prevent damage to the device during measurement.

- Effective measurement modes:
  - I/V Sweep measurement
  - Multi Channel I/V Sweep measurement
  - I/V List Sweep measurement
  - C-V Sweep measurement
- Stop conditions:
  - The output reaches voltage compliance or current compliance
  - A measurement value exceeds the specified measurement range
  - An SMU oscillates
  - An error is caused in the MFCMU

This automatic sweep abort function is enabled by using the Sweep [xxxx] status field on the Measurement Setup screen of the Classic Test definition.

- Sweep [CONTINUE AT ANY] status: Sets the abort function OFF. Not available if the power compliance has been set.
- Sweep [STOP AT ANY ABNORMAL] status: Sets the abort function ON.
- Sweep [SKIP AT ANY ABNORMAL] status: Sets the abort function ON. However if a stop condition is detected during a subordinate sweep measurement, this function stops the VAR1 sweep once, changes the VAR2 step value, and continues the sweep measurement. This mode is not available for Desktop EasyEXPERT and for I/V List Sweep and C-V Sweep measurements.

### When abort occurs

After measurement is aborted, the source forces the following value.

- Sweep source: Start value
- Pulsed source or Pulsed sweep source: Pulse base value

## Standby Function

This function forces the specified DC bias outputs (standby outputs) before starting or after stopping a measurement.

### Standby Channels

Standby channels are SMU or UHVU that force the standby output. Standby channels maintain the specified DC bias output when the standby mode is ON and in the non-measurement state.

Standby channel cannot be set for CMU, GNDU, and SMU connected to the atto sense and switch unit (ASU) which makes the path to the AUX port.

To specify the measurement resource connected to the selector (N1258A, N1259A-300, N1265A) or N1266A expander, the channel must be set to the Default output in the Configuration window.

See “Standby Channel Definition” on page 1-83 to define the standby channels and set the standby mode.

### Standby State

Standby state is the state before starting or after stopping a measurement, in which only the standby channels perform DC bias outputs. The other channels will be in the idle state, which is 0 V output, 100  $\mu$ A compliance, SMU filter OFF, and series resistor OFF. When the standby mode is OFF and in the non-measurement state, all channels will be in the idle state.

When the B1500 is in the standby state, the measurement trigger stops the standby state, so the standby channels stop the standby output and the measurement channels perform a measurement (measurement state). After the measurement, the standby channels start the standby output (standby state). During this series of operations, the state is never changed to the idle state. See Figure 4-20 for an example operation.

If the bias hold function is enabled for the standby channel, after the measurement, the channel forces the bias output set by the bias hold function. After the bias hold period, the channel starts the standby output. During this series of operations, the state is never changed to the idle state.

## Output Sequence of Standby Channels

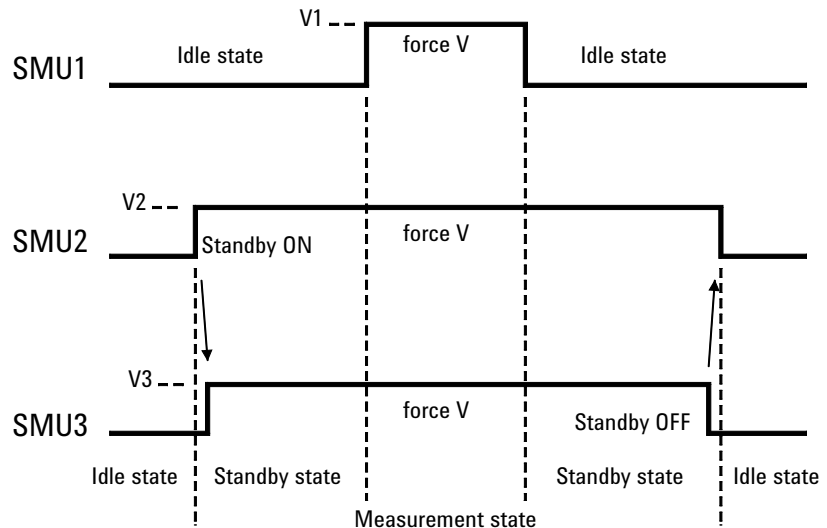
The standby channels will start to output in order, from the top to bottom, as listed in the Standby Channel Definition on the Standby Channel Definition window. The order of bias stop is bottom to top.

Figure 4-20 shows the standby and measurement operations for the following example setup. This example assumes that the channel setup is defined in the Standby Channel Definition as follows in the order show, and that the output value in the standby state is the same as the output value in the measurement state.

- SMU1: Voltage source, non-standby channel
- SMU2: Voltage source, standby channel
- SMU3: Voltage source, standby channel

Figure 4-20

### Standby Function



## To Use Standby Function

To start the standby output, click the Standby ON button on the Standby Channel Definition window. However, if no channel has been defined as the standby channel, the standby mode cannot be turned ON. In the standby state, the Standby ON indicator appears at the bottom of the EasyEXPERT main screen. The Standby Channel Definition window is opened by clicking the Standby button on the main screen.

To stop the standby output, click the Standby OFF button on the Standby Channel Definition window.

In the standby state, you can change the setting parameters for non-standby channels without changing the operation state. However, if you modify the settings for the standby channels, the operation state is changed from the standby state to the idle state.

In the standby state, the following functions are set to OFF.

- SMU filter
- Series resistor

In the standby state, the following functions cannot be performed.

- Diagnosis
- Self-test
- Self-calibration (SMU calibration and CMU calibration)
- SMU zero offset measurement
- ASU connection path change

In the standby state, self-calibration is not performed even if auto calibration is set to ON. Also the state cannot be changed to the standby state while self-calibration is being performed.

When the workspace is opened, it will be in the idle state.

If the workspace or EasyEXPERT is closed while the standby mode is ON, the mode is automatically changed to OFF in the closing process.

If Windows session is stopped while the standby mode is ON, the mode is not changed to OFF in the closing process.

---

## Bias Hold Function

This function is used to keep source output after measurement. Source modules apply the specified bias between measurements in a quick test or application test that defines some classic test setups, or a repeat measurement. And the source modules change the output value and the unused modules are disconnected when the next measurement is started.

### Output channels and output value

If the bias hold function is OFF, the channel stops the output immediately after measurement.

If the function is ON, the channel continues the output as shown below.

*I/V Sweep measurement:*

- VAR1 and VAR1' output channels: Sweep start or stop value
- VAR2 output channel: Sweep stop value
- CONST output channels: Output value
- SMU pulse output channels: Base value

*C-V Sweep measurement:*

- Bias sweep output channel: Sweep start or stop value
- CONST output channels: Output value

*I/V-t Sampling measurement:*

- Constant source output channels: Base or source value

*SPGU output:*

- SPGU keeps the output as shown in Table 4-6.

### Period of bias hold

- From the end of a measurement to the end of a single measurement
- From the end of a measurement to the start of the next measurement during a quick test, application test, or repeat measurement

### Reset of source output

The source output is reset after a quick test, application test, repeat measurement, or single measurement. The SMU is set to 0 V output and 0.1 mA compliance. The MFCMU is set to 0 V DC bias and 0 V oscillator level.

Function Details  
Bias Hold Function

**Setup method**

This function can be set by using the Advanced Setup window.

- Bias hold after measurement  
Set the function ON (enable) or OFF (disable, default setting).
- Output value after measurement  
Select the bias output value START (default) or STOP for the I/V or C-V sweep measurement, or BASE or SOURCE (default) for the sampling measurement. This is invalid for the VAR2, CONST, and constant source for C-V Sweep.

**Table 4-6**

**SPGU Channel Output after Execution when Bias Hold is ON**

<b>SPGU output operation mode</b>	<b>SPGU output</b>
FREE RUN	Continues the pulse output in the VPULSE mode or the sequence output in the ALWG mode. SPGU trigger reset is performed at the beginning of the simultaneous sampling measurement.
COUNT	Continues the base voltage output in the VPULSE mode or the initial voltage output in the ALWG mode. SPGU channel initialization is performed at the beginning of the next measurement.
DURATION	

---

## Current Offset Cancel

This function subtracts the offset current from the current measurement raw data, and returns the result as the measurement data. This function is used to compensate the error factor (offset current) caused by the measurement path such as the measurement cables, manipulators, or probe card.

The offset cancel is not available for measurement over 100 nA range. For 10 nA range measurement, the offset cancel is performed by using the 1 nA range offset value. For measurement by a range less than 10 nA, the offset value measured by each range is used. See Table 4-7 for the effective measurement range, the allowable offset value, and the module type of the measurement channel.

### Enabling offset cancel function

You can perform the offset current measurement and enable the offset cancel function as shown below.

1. Click the Calibration button on the main screen to open the Calibration window. See “Calibration” on page 1-58.
2. Specify the channels to perform the offset current measurement on the SMU Zero Cancel tab screen of the Calibration window. To specify the channel, check the check box to the left of the Name field.
3. To enable the 1 pA range offset current measurement of SMU+ASU, check the Full Range check box. Can be set for each channel.
4. Set the integration time for the offset measurement to the Integration Time field. Available value: 1 PLC (power line cycle) to 100 PLC, initial setting 16 PLC. Can be set for each channel.

Set the value longer than the integration time for the measurement of the device under test. Otherwise, the offset cancel will not be performed.

5. Click the Measure... button. A dialog box is opened.

Follow the instruction of the dialog box, and perform the offset measurement. After the measurement, the offset cancel function is automatically enabled.

Open the measurement terminals at the cable end of the device side during the offset current measurement.

Function Details  
Current Offset Cancel

---

**NOTE**

Except for HVSMU, the high resolution A/D converter is used for the offset current measurement.

To disable the offset cancel function, remove the check from the check box to the left of the Name field.

---

**Table 4-7**

**Measurement range, allowable offset value, and module type**

<b>Measurement range</b>	<b>Allowable offset value</b>	<b>Module type</b>
1 nA	$\pm 10$ pA	SMU+ASU, HRSMU, MPSMU, HPSMU, HVSMU
100 pA	$\pm 1$ pA	SMU+ASU, HRSMU
10 pA	$\pm 400$ fA	SMU+ASU, HRSMU
1 pA	$\pm 100$ fA	SMU+ASU

If the offset value for a measurement range of a channel exceeds the allowable value, the offset data is not effective in this range and lower for this channel. Then, if the ranging mode setting of this channel covers this range or lower, the offset cancel is not performed for any data measured by this channel.



---

## SMU CMU Unify Unit

Only for the B1500A. The SMU CMU Unify Unit (SCUU) is available for the B1500A installed with the multi frequency capacitance measurement unit (MFCMU). The MFCMU must be installed in the slot N (integer, 3 to 10) and the SMUs must be installed in the slot N-1 and N-2. See Agilent B1500A *User's Guide* to install the SCUU properly. The SCUU cannot be connected to the high power SMU (HPSMU).

The Guard Switch Unit (GSWU) controls the connection of the CMU High and Low guard lines (CMU return). The connection will be broken when using the SMU and made when using the MFCMU.

The SCUU can switch the measurement resources, MFCMU or SMU, and can configure four setups shown in Figure 4-21. When the B1500A is turned on, no measurement resource is connected to the output. The SCUU has input connectors, four coaxial and four triaxial, connected to the MFCMU and SMU directly or by using the dedicated SCUU cable.

The SCUU front panel LEDs indicate the following conditions of the SCUU.

- |                          |   |
|--------------------------|---|
| <b>SMU</b>               | SMU in the slot N-1/N-2 can be used.                    |
| <b>CMU Internal Bias</b> | MFCMU can be used. CMU internal DC bias source is used. |
| <b>CMU SMU Bias</b>      | MFCMU can be used. SMU is used as DC bias source.       |

The SCUU has output connectors, four triaxial, that must be connected to the DUT interface such as manipulators and probe card. When the measurement resource is the SMU, they are the Force1/Sense1 and Force2/Sense2 connectors for the Kelvin connection. And when the resource is the MFCMU, the CMU High/Low are connected to the Force1/Force2 respectively, and the Sense1/Sense2 are not used.

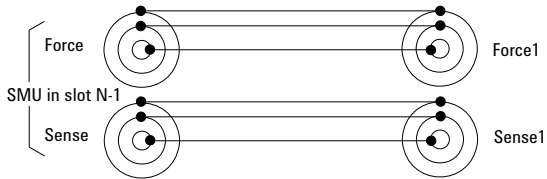
To realize accurate impedance measurements, connect the CMU return terminals together. The CMU return terminal means the shield end of the outer shield conductor extended to the DUT interface. The shield end must be opened as close as possible to the DUT. See Figure 4-14 on page 4-26.

The GSWU has one connector and two terminals. The connector must be connected to the SCUU by using the SCUU cable. And the terminals must be connected to the CMU High/Low guard lines at the shield ends of the DUT interface by using a wire.

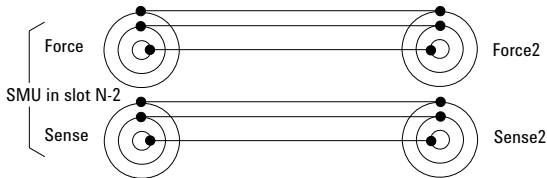
Function Details  
 SMU CMU Unify Unit

Figure 4-21 SCUU Input/Output Connections

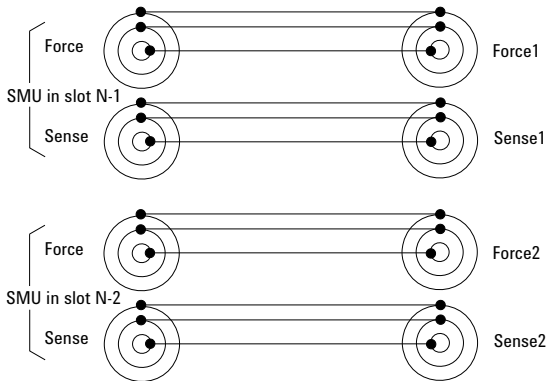
Setup 1: CMH/Force1/Sense1: SMU (slot N-1)  
 CML/Force2/Sense2: Open



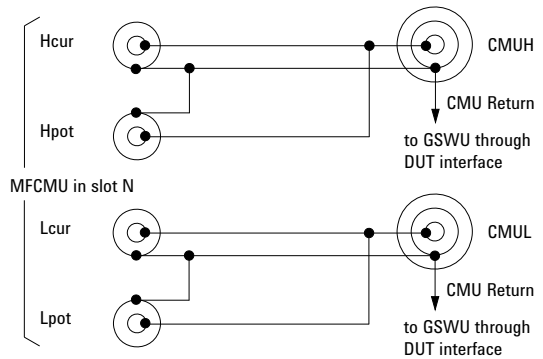
Setup 2: CMH/Force1/Sense1: Open  
 CML/Force2/Sense2: SMU (slot N-2)



Setup 3: CMH/Force1/Sense1: SMU (slot N-1)  
 CML/Force2/Sense2: SMU (slot N-2)



Setup 4: CMH/Force1: CMUH (slot N)  
 CML/Force2: CMUL (slot N)  
 Sense1/Sense2: Open



**NOTE**

Before using SCUU

Instrument connection must be done before the B1500A is turned on.

## Atto Sense and Switch Unit

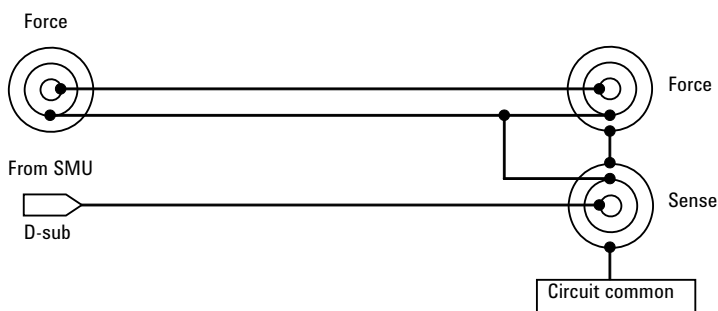
Only for the B1500A. To use the Atto Sense and Switch Unit (ASU), it must be connected to the SMU which is connected together when the calibration is performed. Using the ASU permits use of the 1 pA range. Also the ASU can switch the measurement resources, SMU or an instrument connected to the AUX input connectors. The ASU can configure two setups shown in Figure 4-22. When the B1500A is turned on, the ASU will be the Setup 1 condition. See Agilent B1500A *User's Guide* to install the ASU properly.

The ASU has input connectors, one triaxial and two coaxial. The triaxial connector must be connected to the SMU Force, and the coaxial connectors must be connected to an instrument. Also the ASU has the control connector to be connected to the SMU by using the ASU cable.

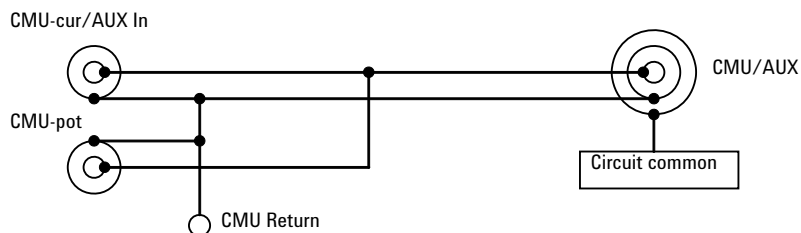
Figure 4-22

### ASU Input/Output Connections

Setup 1: AUX Off ( SMU input to ASU output )



Setup 2: AUX On ( CMU/AUX input to ASU output )



The ASU has the following status indicators (LED).

**SMU**                      The SMU can be used (Setup 1).

**AUX**                      The AUX instrument can be used (Setup 2).

## Function Details

### Atto Sense and Switch Unit

The ASU has output connectors, two triaxial, that must be connected to the DUT interface such as manipulators and probe card. When the measurement resource is the SMU, they are the Force/Sense connectors for the Kelvin connection. And when the resource is not SMU, the CMU/AUX is connected to the Force, and the Sense is not used.

If you connect the MFCMU or the LCR meter with the four-terminal pair configuration, two ASUs are required. Then the CMU-pot/CMU-cur connectors of a ASU must be connected to the Hpot/Hcur of the instrument. And the other one must be connected to the Lpot/Lcur in like manner.

To realize accurate impedance measurements using the four-terminal pair measurement method, connect the CMU Return terminals of ASUs together. See “Four-Terminal Pair Configuration” on page 4-24 and Figure 4-14 on page 4-26.

---

#### NOTE

##### To use 1 pA range

The measurement channel connected to the ASU (Atto Sense and Switch Unit) supports the 1 pA range. To use the 1 pA range, set the 1 pA fixed range or the 1 pA limited auto ranging.

The B1500A automatically performs the compensation of the data measured by the 1 pA range and returns the compensated data. You can use either the pre-stored offset data or the pre-measured offset data.

To measure the offset data, perform the self-calibration before starting the measurement for a DUT. This offset data is temporarily memorized until the B1500A is turned off.

---

#### NOTE

##### Before using ASU

ASU connection must be done before the B1500A is turned on.

The combination of ASU and SMU is important. The B1500A can work with wrong combination, but it cannot satisfy its specifications. The specifications are guaranteed for the correct combination which the performance verification was performed. See “ASU” on page 1-66 and check if the combination is correct.

The CMU-pot connector is for the MFCMU or the LCR meter that uses the four-terminal pair. For the other instrument, connect a BNC open connector or nothing to there. The line will be internally connected to the CMU-cur/AUX-in line.

It is unavailable to use the series resistor in the SMU connected to the ASU.

---

## SMU/PG Selector

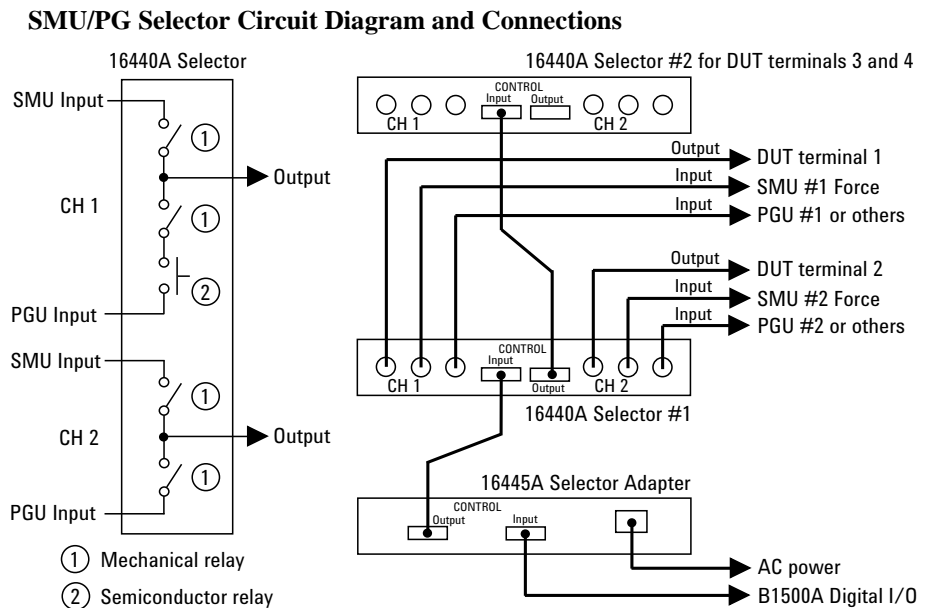
Not available for the B1505A. Agilent 16440A SMU/PGU selector (B1500A-A04) provides the built-in switching capability to select a measurement path connected to the Output port. See Figure 4-23 for the simplified circuit diagram of the selector. The selector has the channels CH 1 and CH 2 which provide the following conditions. It can be controlled by using the SMU/PG Selector tab screen of the Configuration window. See “SMU/PG Selector” on page 1-69.

**CH 1** Open (no connection), SMU connect, PGU connect, and PGU open

**CH 2** Open (no connection), SMU connect, and PGU connect

The PGU open is available only for the CH 1, and is made by keeping the PGU side mechanical relay close and opening the semiconductor relay. This is effective for frequent switching applications because the semiconductor relay is more durable than mechanical relays. The PGU open can be set by using the Advanced Setup window of the Measurement Setup. See “Advanced Setup Window” on page 2-53.

Figure 4-23



The selector must be connected to the B1500A through the Digital I/O connector and the Agilent 16445A Selector Adapter. Up to two selectors can be used by connecting the selector control cable between the CONTROL Output connector of the first selector and the CONTROL Input connector of the second selector.

## Module Selector

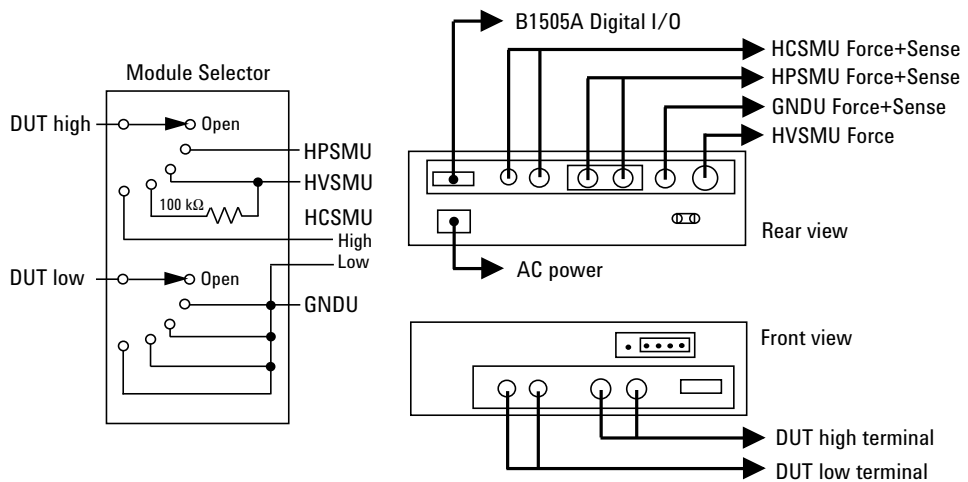
Available for the B1505A. The Agilent N1258A Module Selector is used to switch the measurement resources (HCSMU or DHCSMU (dual HCSMU), HPSMU, and HVSMU) connected to DUT (device under test). The Input ports should be connected to the HCSMU or DHCSMU, GNDU, HPSMU, and HVSMU. At least, two SMU modules must be connected to the Input ports. And the Output port must be connected to the DUT interface.

For the packaged devices, use the Agilent N1259A test fixture which can install the module selector.

The module selector can be controlled by using the Module Selector tab screen of the Configuration window. See “Module Selector” on page 1-69.

Figure 4-24

### Module Selector Circuit Diagram and Connections



### CAUTION

If dual HCSMU (DHCSMU) is used with the module selector, the maximum current must be  $\pm 30$  A to prevent the module selector from performance degradation and failure.

## Ultra High Current Expander/Fixture

Available for the B1505A. The N1265A is a test fixture which contains the current expander to enable 500 A or 1500 A (option N1265A-015) output and measurement, and contains the selector to switch the measurement resource connected to the DUT.

The current expander is used to configure the ultra high current unit (UHCU).

The selector is used to switch the measurement resource connected to the DUT. The measurement resource will be the UHCU, HVSMU, HP/MPSMU, or GNDU. See Figure 4-26.

The selector output can be extended to your prober station by using the N1254A-524 ultra high current prober system cable. This cable supports current up to 500 A.

The N1265A also provides the connection paths for UHVU, MFCMU via bias-tee, and HP/MP/HC/DHCSMU.

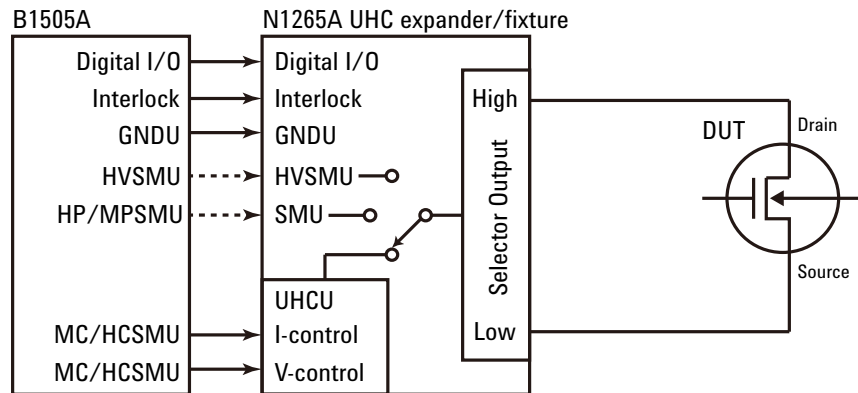
### Ultra High Current Unit

The ultra high current unit (UHCU) can be configured by using the N1265A and two MC/HCSMU modules as shown in Figure 4-25. The UHCU output can appear on the Selector Output by controlling the built-in selector.

The ultra high current expander/fixture can be controlled by using the UHC Expander / Fixture tab screen of the Configuration window. See “UHC Expander / Fixture” on page 1-72.

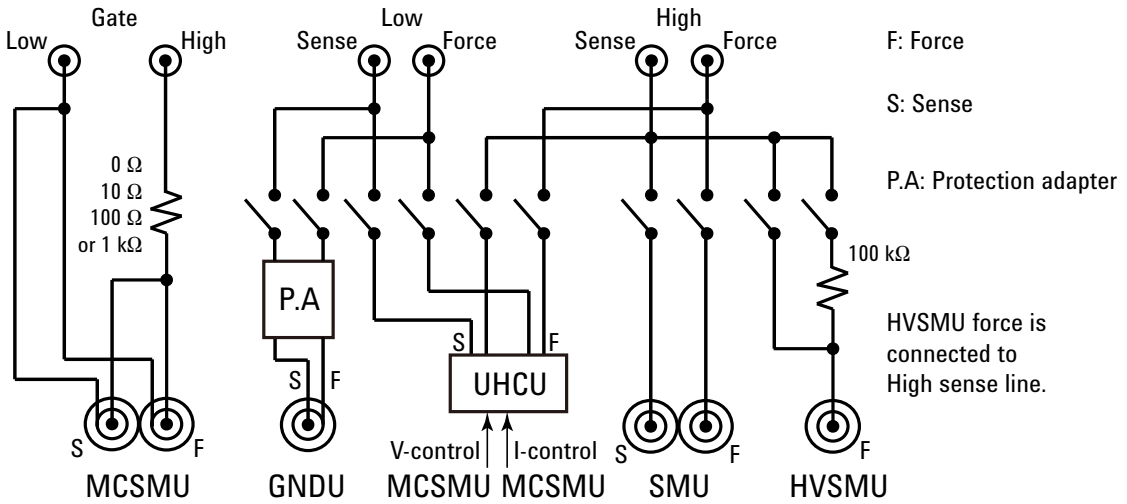
Figure 4-25

### To Configure UHCU



Function Details  
Ultra High Current Expander/Fixture

Figure 4-26 N1265A Built-in Selector Simplified Internal Connections





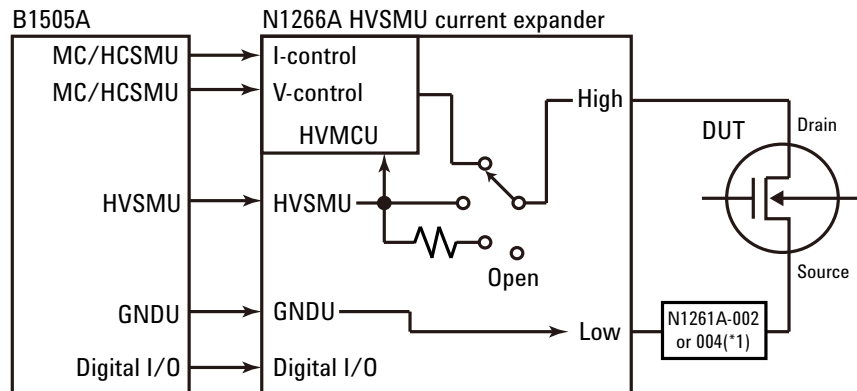
## HVSMU Current Expander

Available for the B1505A. The N1266A is a current expander for HVSMU. The N1266A is used to configure the high voltage medium current unit (HVMCU) with the HVSMU module and two MC/HCSMU modules as shown in Figure 4-27. Selector is initially installed for switching the HVSMU or the HVMCU connected to the DUT.

The HVSMU current expander can be controlled by using the HVSMU Current Expander tab screen of the Configuration window. See “HVSMU Current Expander” on page 1-75.

Figure 4-27

### To Configure HVMCU



\*1: N1261A-002/004 protection adapter is required for connecting prober.  
Not required for connecting N1258A, N1259A, N1259A-300, or N1265A.

---

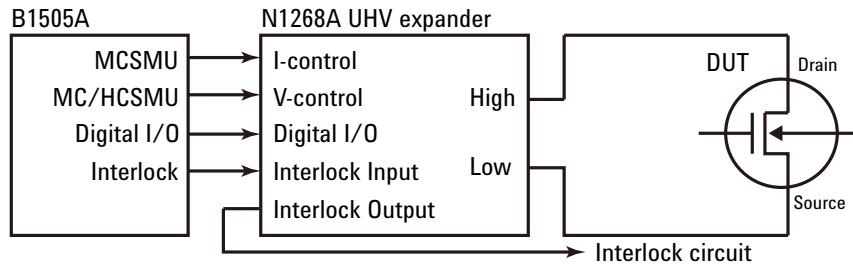
## Ultra High Voltage Expander

Available for the B1505A. The N1268A is a voltage expander to enable 10 kV output and measurement. The N1268A is used to configure the ultra high voltage unit (UHVU) with two MC/HCSMU modules as shown in Figure 4-28.

The ultra high voltage expander can be controlled by using the UHV Expander tab screen of the Configuration window. See “UHV Expander” on page 1-77.

**Figure 4-28**

### To Configure UHVU



---

## SMU Ranging Mode

Agilent B1500 provides the following operation modes for the measurement range and the output range. The ranging mode must be set for each channel.

- “Auto Ranging”
- “Limited Auto Ranging”
- “Fixed Range”
- “Compliance Range”
- “Enhanced Auto Ranging for Current Measurement”

### To Set Ranging Mode

The following points must be noted when setting the ranging mode.

#### For measurement channels

- If you do not specify the ranging mode, the auto ranging mode is set automatically.
- For the channels that perform Vf/Vm (voltage force and voltage measurement) or If/Im (current force and current measurement), the measurement range is always the same as the output range.
- For measurement with pulse output, the measurement channel uses the compliance range or the specified fixed range. The channel cannot perform the auto ranging operation or the limited auto ranging operation.
- For the SMU connected to the atto sense/switch unit (ASU), the 10 pA range is the minimum for the auto ranging operation. To use the 1 pA range, select the 1 pA limited auto ranging mode or the 1 pA fixed range mode.

#### For output channels

- The auto ranging mode and the limited auto ranging mode are available for the output channels.
- When the voltage output range or the voltage compliance range is changed, the channel forces 0 V at that moment.

## Auto Ranging

Range changing is performed as below.

### For measurement channels

The measurement channel automatically searches for and measures at the range that provides the best resolution as follows:

- For current measurement, the measurement channel uses the present range if the following formula is satisfied; otherwise, the channel changes the range.

$$\text{Range} \times 0.1 \leq \text{Measurement value} \leq \text{Range} \times X/100$$

where X=115, or 100 at the maximum range.

See also “Enhanced Auto Ranging for Current Measurement” on page 4-58.

- For voltage measurement, the measurement channel uses the present range if the following formula is satisfied; otherwise, the channel changes the range.

$$\text{Next lower range value} \leq \text{Measurement value} \leq \text{Range} \times X/100$$

where X=110, or 100 at the maximum range.

### For output channels

The output channel automatically searches for and forces at the lowest range that covers the output value as follows:

- For staircase sweep output, the output channel uses the lowest range that covers the start and stop values. However, it uses the lowest range that covers the step output value for the logarithmic current sweep. For the voltage sweep that sets the power compliance, the channel uses the lowest range (20 V range or more) that covers the step output value.
- For pulsed bias output, the output channel uses the lowest range that covers the base and peak values.
- For pulsed sweep output, the output channel uses the lowest range that covers the base, start, and stop values.
- For quasi-pulsed output and search source output, the output channel uses the lowest range that covers the start and stop values.

The maximum output value is as follows:

- Maximum current output value =  $\text{Range} \times X/100$

where X=115, or 100 at the maximum range.

- Maximum voltage output value = Range value

## Limited Auto Ranging

Limited auto ranging is similar to auto ranging. However, limited auto ranging does not use the range(s) less than what you specified. For example, if you select 10 mA limited auto ranging, the measurement module does not use the 1 mA range or less. Consequently, the measurement time for limited auto ranging is less than for auto ranging.

## Fixed Range

The channel uses the specified range only.

For the current measurement channel, if you specify a range higher than the minimum range that covers the compliance value, the measurement channel uses this range (compliance range).

## Compliance Range

The compliance range is effective for the  $V_f/I_m$  (voltage output and current measurement) or  $I_f/V_m$  (current output and voltage measurement) channel. The measurement channel automatically selects and uses the lowest range that covers the specified compliance value.

For the measurement channel that performs constant voltage or current output, you can set the compliance range.

For the measurement channel that performs sweep output, you can set the power compliance value. If you set the power compliance value, the channel uses the lowest range that covers the lower value of either the specified compliance value or the compliance value given by the following formula.

Current compliance = Specified power compliance value / Step voltage

Voltage compliance = Specified power compliance value / Step current

For details about setting compliance, refer to “SMU Compliance” on page 4-59.

---

**NOTE**

---

Range change is performed when the compliance value is set, not when the measurement is performed.

## Enhanced Auto Ranging for Current Measurement

Expanded functions are available for the auto ranging operation of the current measurement channel. To use the functions, set *Range Change Rule*. See “SMU Range Setup Window” on page 2-48. *Rate* value defines the boundary of ranging.

- *Range Change Rule*=BY FULL RANGE:  
Performs normal auto ranging operation. If a condition shown below is satisfied, the range changes immediately, even during measurement.  
$$\text{Measurement value} > \text{Range} \times X/100$$
$$\text{Measurement value} \leq \text{Range} \times 0.1$$
where  $X=115$ , or  $100$  at the maximum range.
- *Range Change Rule*=GO UP AHEAD:  
If the following condition is satisfied, the range changes up after measurement.  
$$\text{Measurement value} \geq \text{current1}$$
- *Range Change Rule*=UP AND DOWN AHEAD:  
If the following condition is satisfied, the range changes down immediately, even during measurement.  
$$\text{Measurement value} \leq \text{current2}$$
  
If the following condition is satisfied, the range changes up after measurement.  
$$\text{Measurement value} \geq \text{current1}$$

where *current1* and *current2* are given by the following formula. Then *rate* must be an integer, 11 to 100.

$$\text{current1} = \text{Range} \times \text{rate}/100$$

$$\text{current2} = \text{Range} \times \text{rate}/1000$$

For example, if the range value is 10 mA and *rate* is 90, they are:

$$\text{current1} = 9 \text{ mA}$$

$$\text{current2} = 0.9 \text{ mA}$$

## SMU Compliance

Compliance is the output limiter to prevent damage to the test device from overcurrent, overvoltage, or overpower. Voltage compliance is for the current output channels, and current compliance is for the voltage output channels.

When a channel reaches compliance, the channel acts as a constant voltage source or a constant current source. The channel keeps the output value when reaching compliance.

Compliance can be set with the same resolution and accuracy as output current or output voltage. For the values, see Agilent B1500A or B1505A *User's Guide*.

### Polarity and Output Area

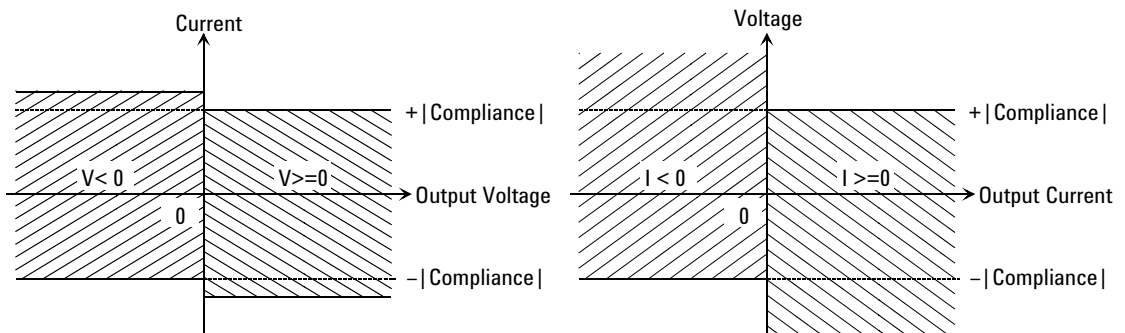
- Polarity of current compliance

The B1500 automatically sets current compliance for both positive and negative polarity. For opposite polarity as the output voltage, the compliance value is increased by an amount that is 2.5 % to 12 % of the range value in the lowest range that covers the specified compliance value. See Figure 4-29.

- Polarity of voltage compliance

The B1500 automatically sets voltage compliance to the same polarity as the output current. There is no compliance for opposite polarity.

**Figure 4-29** Relation of Compliance and Output



## Power Compliance

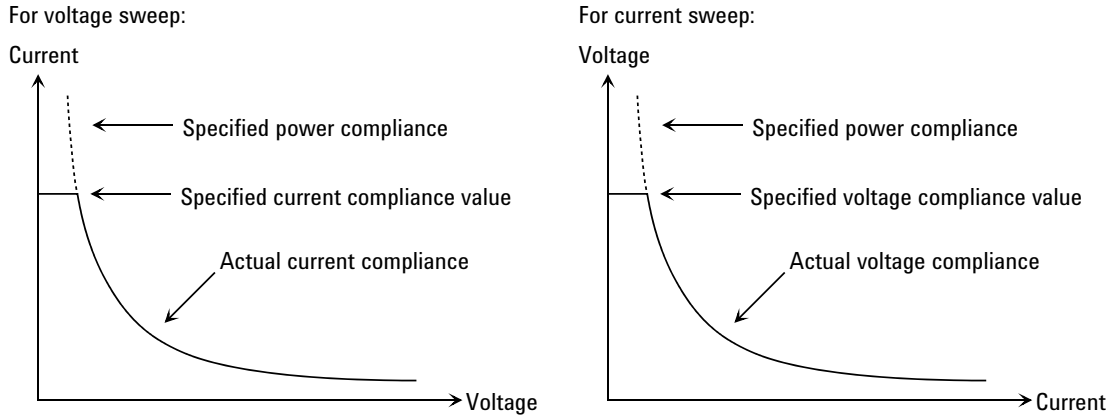
In addition to V compliance or I compliance, you can set power compliance for the staircase sweep source. Power compliance is not available for HVSMU.

If you specify power compliance, SMU changes the current or voltage compliance value every sweep step. The value is the lower value of either the specified compliance value or the compliance value given by the following formula. See Figure 4-30.

Current compliance = Specified power compliance value / Step voltage

Voltage compliance = Specified power compliance value / Step current

**Figure 4-30 Power Compliance Output Area**



### Available values

The following values are available for the power compliance:

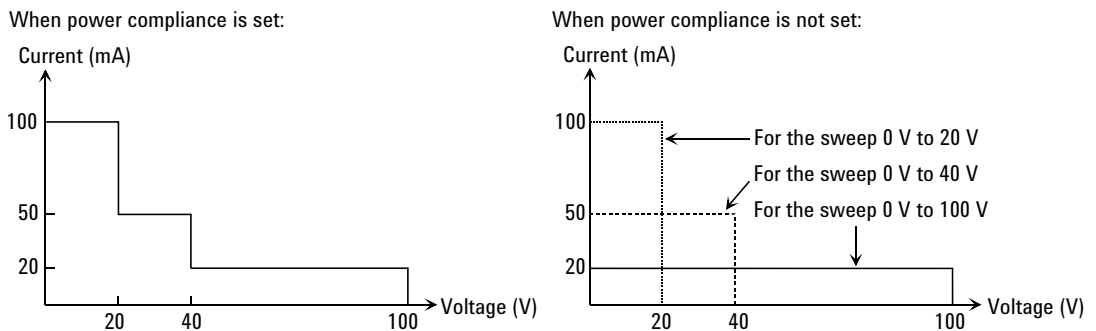
<b>MPSMU/HRSMU</b>	1 mW to 2 W
<b>HPSMU</b>	1 mW to 20 W
<b>HCSMU</b>	1 mW to 40 W (DC), 1 mW to 400 W (pulse)
<b>DHCSMU</b>	1 mW to 80 W (DC), 1 mW to 800 W (pulse)
<b>MCSMU</b>	1 mW to 3 W (DC), 1 mW to 30 W (pulse)
<b>UHCU</b>	1 mW to 7.5 kW (pulse, 500 A range), 1 mW to 22.5 kW (pulse, 1500 A range)
<b>UHVU</b>	1 mW to 100 W (DC), 1 mW to 200 W (pulse)
<b>HVMCU</b>	1 mW to 600 W (pulse, 2200 V range), 1 mW to 900 W (pulse, 1500 V range)



**Conditions after compliance** When a channel reaches compliance, the staircase sweep source automatically stops the sweep and forces the start value. After reaching the compliance, the measurement data will be “NaN”.

**SMU output values** If you specify power compliance, the SMU changes the current or voltage compliance every sweep step. Hence, the SMU can apply voltage or current with the maximum power. Figure 4-31 shows the differences between the output with power compliance and the output without power compliance, for MPSMU.

**Figure 4-31 Allowable Current in Voltage Sweep (0 V to 100 V, MPSMU)**



**NOTE**

If you specify power compliance, the measurement time increases slightly because the B1500 adjusts the compliance for every sweep step.

**To Set Compliance**

The following points must be noted when setting the compliance.

- If the current compliance value is too low, the SMU will take a long settling time.
- The SMU will reach voltage compliance if the SMU cannot force the specified current for the following reasons:
  - Over current for test device.
  - Another channel has reached compliance.
  - Current measurement data of another channel has reached the limit for the present measurement range.

## SMU Pulse

SMU can apply voltage or current pulse. This section describes the pulse setup parameters. The parameters are different by the number of pulse output channels as shown in Figure 4-32.

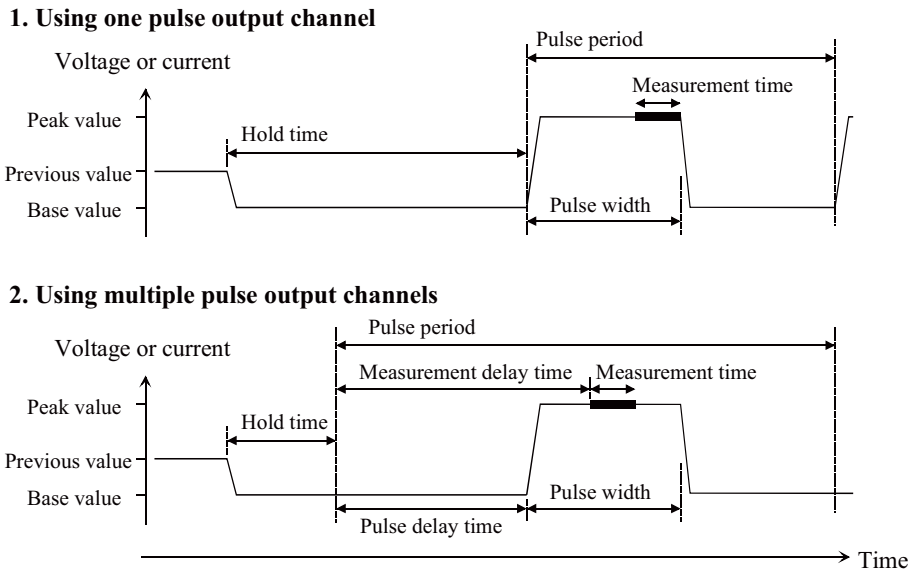
### To Set Pulse Output

The following points must be noted when setting the pulse output.

- Cannot use the 1 nA output range and lower ranges.
- Cannot use multiple pulse output channels for the tests other than the Multi Channel I/V Sweep.
- Cannot use multiple measurement channels for the tests other than the Multi Channel I/V Sweep.
- The measurement channel uses the compliance range. The channel cannot perform the auto ranging operation or the limited auto ranging operation.

Figure 4-32

#### SMU Pulse Setup Parameters



## Pulse Setup Parameters

The pulse setup parameters and the available values are shown below.

<b>Hold time</b>	0 s to 655.35 s, 10 ms resolution
<b>Pulse period</b>	5 ms to 5 s, 100 $\mu$ s resolution
<b>Pulse width</b>	HR/MP/HPSMU: 500 $\mu$ s to 2 s, 100 $\mu$ s resolution HCSMU / dual HCSMU: 50 $\mu$ s to 2 s, 2 $\mu$ s resolution. Maximum 1 ms and duty ratio $\leq$ 1 % for using 20 A range or 40 A range. MCSMU: 10 $\mu$ s to 2 s, 2 $\mu$ s resolution. Maximum 100 ms and duty ratio $\leq$ 5 % for using 1 A range. UHCU: 10 $\mu$ s to 1 ms and duty ratio $\leq$ 0.4 % for 500 A range, 10 $\mu$ s to 500 $\mu$ s and duty ratio $\leq$ 0.1 % for 1500 A range, 2 $\mu$ s resolution. UHVU: 100 $\mu$ s to 2 s, 2 $\mu$ s resolution. Maximum 1 ms for using 100 mA range. HVSMU: 500 $\mu$ s to 2 s, 2 $\mu$ s resolution HVMCU: 10 $\mu$ s to 100 $\mu$ s, 2 $\mu$ s resolution. Maximum 1 ms for using 2200 V / 110 mA range.
<b>Pulse delay time</b>	0 to Pulse period – Pulse width, 2 $\mu$ s resolution 0 s for using HR/MP/HPSMU
<b>Measurement delay time</b>	2 $\mu$ s to Pulse period – Measurement time – 2 ms, 2 $\mu$ s resolution
<b>Measurement time</b>	HR/MP/HPSMU: 16 $\mu$ s to 20 ms, 2 $\mu$ s resolution HC/MCSMU: 2 $\mu$ s to 20 ms, 2 $\mu$ s resolution HVSMU: 6 $\mu$ s to 20 ms, 2 $\mu$ s resolution See “Integration Time” on page 4-65.

The setup values must satisfy the following conditions.

- Pulse period  $\geq$  pulse delay + width + 2 ms, for pulse delay + width  $\leq$  100 ms
- Pulse period  $\geq$  pulse delay + width + 10 ms, for pulse delay + width  $>$  100 ms

## Function Details

### SMU Pulse

- Pulse width must be the same for all of the HR/MP/HPSMU pulse output channels. If a different value is entered, the longest value is set.

Pulse period = AUTO sets the longest one of the followings automatically.

- Maximum period given by the pulse width and the duty ratio
- Pulse period = 5 ms, for pulse delay + width  $\leq$  3 ms
- Pulse period = pulse delay + width + 2 ms, for 3 ms < pulse delay + width  $\leq$  100 ms
- Pulse period = pulse delay + width + 10 ms, for 100 ms < pulse delay + width

Measurement delay time = AUTO sets the optimum value automatically so that the measurement is completed when the transition from peak to base is started by the pulse output channel which starts the transition at first.

## SMU Measurement Time

Measurement time depends on integration time, measurement range, and other measurement conditions, and can be expressed by the following formula:

$$\text{Measurement time} = \text{Integration time} + \text{Overhead time}$$

Integration time is the time required for measurement, and does not include such factors as range changing or data compensation, which would be the overhead time.

### Integration Time

Integration time is the time required to get measurement data. For accurate and reliable measurement, integration time should be increased. Integration time can be set for each type of A/D converter (ADC) used by the measurement channel.

- High-speed ADC

Installed in each measurement module. For high speed measurement. Sets the number of averaging samples for getting one measurement data by setting the *Mode* and the *Factor* value. Only PLC and TIME are available for SMU pulse.

**AUTO**      *Number of averaging samples = Factor × reference*

*Factor = 1 to 1023*

where *reference* is the number of averaging samples automatically set by the B1500; this cannot be changed.

**MANUAL**      Specify the number of averaging samples (*Factor*).

*Factor = 1 to 1023*

**PLC**      *Number of averaging samples = Factor × 128*

*Factor = 1 to 100*

The B1500 gets 128 samples in a power line cycle. Hence, the *Factor* value is equal to the number of power line cycles.

**TIME**      Specify the actual measurement time (*Factor*).

*Factor = 2 μs to 20 ms, 2 μs step*

For HR/MP/HPSMU, it performs the operation of MANUAL. Then the number of averaging samples is automatically set to the following value.

## Function Details

### SMU Measurement Time

$$\text{Number of averaging samples} = \text{Factor} / T_m$$

$T_m$ : Required time for one sample measurement

If  $\text{Factor} / T_m < 1$ , *Number of averaging samples* = 1.

- High-resolution ADC

For high accuracy and high resolution measurement. This ADC is mounted on the mainframe, and used by the multiple modules alternately. Sets the integration time for getting one measurement data by setting the *Mode* and the *Factor* value. The high-resolution ADC cannot be used for the measurements using SMU pulse.

**AUTO**      *Integration time* = *Factor* × *reference*

$$\text{Factor} = 1 \text{ to } 127$$

where *reference* is the integration time automatically set by the B1500; it cannot be changed.

**MANUAL**      *Integration time* = *Factor* × 80 μsec

$$\text{Factor} = 1 \text{ to } 127$$

**PLC**      *Integration time* = *Factor* / *power line frequency*

$$\text{Factor} = 1 \text{ to } 100$$

The *Factor* value means the number of power line cycles.

### ADC Zero Function

The ADC zero function is available for the high-resolution A/D converter (ADC) and not available for the high-speed ADC. This function measures the internal offset of the high-resolution ADC and compensates it automatically. The measurement data displayed on the LCD will be the data after compensation.

---

### NOTE

---

This function is especially effective for the low voltage measurement. Disable the function in cases where measurement speed is more important than measurement accuracy. This approximately halves integration time.

## Overhead Time

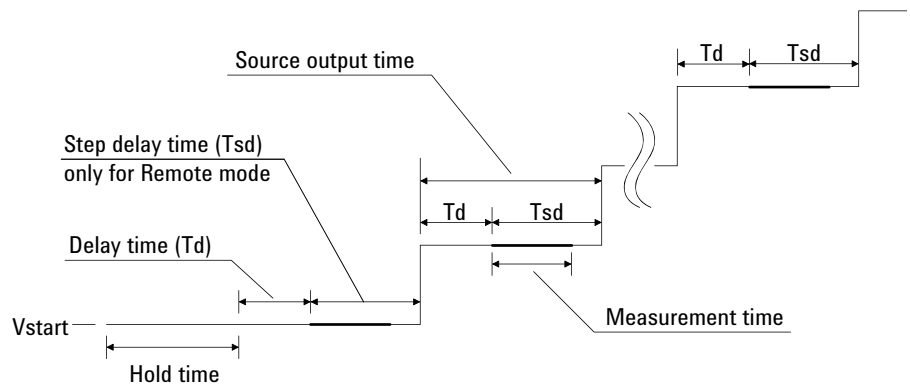
The overhead time is the time required for range changing and so on. This time depends on the measurement condition, and cannot be specified. Major elements of the overhead time are:

- Range changing time during measurement (when measurement ranging mode is set to auto or limited auto)
- Range changing time at measurement start (when the measurement range is less than the compliance value)

## To Specify Source Output Time

You cannot specify the measurement time. However, in sweep measurement mode, you can specify a source output time that includes the measurement time. To specify the source output time, set the delay time and the step delay time. The step delay time is defined as the time from the measurement trigger up to the trigger for changing the source output value. See the figure below. The step delay time must cover the measurement time to keep the source output time constant.

The EasyEXPERT does not support the step delay time. It is available only for the GPIB remote mode.



## Wait Time

Wait time is the time after starting source output until changing the source output value or until starting measurement. You can set the source wait time and the measurement wait time individually.

The source wait time is the time the source channel always waits after it starts output until it changes the output value. The measurement wait time is the time the measurement channel always waits after the source channel starts output until the measurement channel starts measurement. Refer to Figure 4-33. The wait time is given by the following formula:

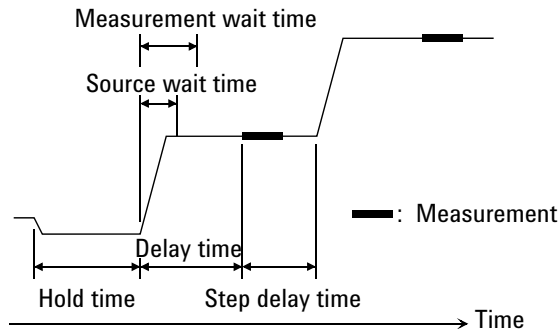
$$\text{wait time} = N \times \text{reference}$$

where *reference* is the wait time the B1500 automatically sets according to the present settings; this wait time cannot change. Available *N* values are 0 to 10, 0.1 step. Initial value is 1.

It is not easy to determine the best wait time. If you specify it too short, the measurement may start before device characteristics stable. If too long, time will be wasted. For measurements of low capacitance or fast response devices, if measurement speed has top priority or is more important than reliability and accuracy, set the wait time shorter. The initial wait time may be too short for measurements of high capacitance or slow response devices. Then set the wait time longer.

Figure 4-33

### Source/Measurement Wait Time



---

#### NOTE

In the GPIB remote mode, if the delay time is longer than the wait time, the wait time can be ignored.

The EasyEXPERT does not support the step delay time.

---



## Multiple Measurement Channels

To use multiple measurement channels, define multiple measurement parameters in the Display Setup screen or the Function Setup screen. Then, measurement sequence, time data, and measurement time for each measurement point are as follows.

- Sequence of measurement execution

Measurement channels perform measurement in the order shown below.

1. Measurement channels which use the high speed AD converter and the FIXED measurement range start measurement simultaneously (parallel measurement).
2. After the parallel measurement, the other measurement channels perform measurement sequentially. The order is top to bottom of the channels defined in the SMU Range Setup screen.

As a special case, for the sweep measurement with multiple pulse outputs and for the linear sampling measurement with *Interval* < 2 ms, all channels start measurement simultaneously.

- Time data

Only the measurement start time of the channel which starts measurement at first is recorded.

- Measurement time

Measurement time is the sum of the measurement time by all channels.

---

## SMU Filter

The filter is mounted on each source/monitor unit (SMU) module. It assures clean source output with no spikes or overshooting. However, using a filter may increase the SMU settling time.

To set the filter, refer to “Advanced Setup Window” on page 2-53.

---

### NOTE

The SMU filter is set to OFF for the standby channels in the standby state.

---

## SMU Series Resistor

The series resistor (approx. 1 M $\Omega$ ) is mounted on each source/monitor unit module (HRSMU, HPSMU, and MPSMU). The series resistor may be used for the device protection, negative resistance measurement, and so on. It depends on the characteristics of test device and measurement environment.

If you use a series resistor, the voltage you set is applied to the near side of the series resistor. Thus, the voltage will be divided by the series resistor and the device under test. The setting value is not the voltage at the device terminal.

To set the series resistor, refer to “Advanced Setup Window” on page 2-53.

The series resistor is not mounted on MCSMU, HCSMU, and HVSMU.

---

### NOTE

#### Restrictions

The series resistor cannot be used for the measurements that use the SMU connected to the Atto Sense and Switch Unit (ASU) or the measurements that use 1 A range of the high power SMU (HPSMU).

The series resistor is set to OFF for the standby channels in the standby state.

---

## Interlock Function

The interlock function is designed to prevent electrical shock when a user touches the measurement terminals. If the Interlock terminal is open, maximum output is limited to  $\pm 42$  V.

To perform high voltage measurement more than  $\pm 42$  V, connect the Interlock terminal to an interlock circuit of test fixture or connector plate. The interlock circuit must have one LED and two mechanical switches placed near the shielding box opening doors. And they must be connected together. For details on how to install the interlock circuit, see Agilent B1500A or B1505A *User's Guide*.

The interlock function works as shown below.

- When the interlock circuit is open, maximum output is limited to  $\pm 42$  V.
- When the interlock circuit is close, source channel can force its maximum output value.
- When the interlock circuit is opened in the high voltage condition more than  $\pm 42$  V, output voltage is set to 0 V immediately.

---

**CAUTION**

---

If the actual output value is not set to 0 V within 120 msec, the B1500 automatically performs the initialization. Then perform self-test. Any module may be defective.

---

**WARNING**

---

**Dangerous voltage of up to the maximum voltage of SMUs may be present at force, guard, and sense terminals if the Interlock terminal is closed.**

## Auto Power Off Function

If an abnormal voltage or current is detected in the B1500, the B1500 automatically turns off to prevent damage.

If this function is activated, the B1500 is turned off, but the Standby switch is left ON. Perform the following steps to turn the B1500 on again.

1. If Windows is running, end all tasks, and shutdown Windows.
2. Open the measurement terminals.
3. Press the Standby switch to set it to the OFF position.
4. Remove the power cable from the B1500 rear panel.
5. Wait for 10 seconds or more.
6. Connect the power cable from the B1500 rear panel.
7. Press the Standby switch to set it to the ON position.

Abnormal voltage or current can be caused by incorrect connections of the B1500 to the test device, excess voltage or current input, momentary power loss, defects of the instrument, and so on.

If no cable is connected to the measurement terminals of the B1500, and the line voltage is correct, the B1500 may have a defect. Contact the nearest Agilent Technologies for assistance.

## Initial Settings

Agilent B1500 is initialized by turning it on, the \*RST command, or the device clear. Initial settings of the B1500 are shown in the following tables.

**Table 4-8 Initial Settings**

	Setup Item	Initial Setting		
Mainframe	Auto calibration	off		
SMU	Output switch	close		
	Filter	on		
	Series resistor, only for HR/MP/HP	off		
	ASU path/indicator	SMU side/enable		
	ASU 1 pA auto range	disable		
	Current measurement range	with pulse	compliance range	
		without pulse	auto	
	Voltage measurement range	with pulse	compliance range	
		without pulse	auto	
	A/D converter	HR/MP/HPSMU: high resolution ADC HC/DHC/MC/HVSMU: high speed ADC		
	ADC integration time	high speed ADC: auto		
		high resolution ADC: auto		
	ADC zero function	off		
	Sweep automatic abort function	off		
	Hold time	0 s		
Delay time	0 s			
Pulse width	0.001 s			
Pulse period	0.01 s			

	Setup Item	Initial Setting
CMU	SCUU path/indicator	open/enable
	Measurement parameter	Cp-G
	Measurement range	auto
	Measurement time	auto
	Open/short/load correction	off
	Phase compensation mode	auto
	Measurement signal frequency	1 k, 2 k, 5 k, 10 k, 20 k, 50 k, 100 k, 200 k, 500 k, 1 M, 2 M, 5 MHz
	Automatic abort function	off
	Hold time	0 s
	Delay time	0 s
SPGU	Operation mode	VPULSE (PG mode)
	Pulse period	1.0 $\mu$ s
	Channel output mode	Primary Pulse, 2-level pulse output
	Pulse source setup	Delay: 0 s Width 100 ns Leading: 20 ns Trailing: 20 ns
	ALWG setup	Cleared
	Pulse switch	Disable
	DUT load impedance	50 $\Omega$
	SPGU trigger output	Disable

Function Details

Initial Settings



---

**5**

**Built-in Programming Tool**

---

## Built-in Programming Tool

This chapter explains the programming tool built into Agilent EasyEXPERT. The programming tool is used to define user functions and analysis functions in the Classic Test definitions and to define the test contents in the Application Test definitions.

This chapter contains the following sections.

- “Variables and Expressions”
- “Built-in Functions”
- “Read-out Functions”
- “Script Program Statements”

## Variables and Expressions

Variables can be used in Agilent EasyEXPERT setup editor for the following applications. User functions and analysis functions can also be used for variables.

- To store the value of the measurement condition input parameter
- To store the value of the measurement result data or analysis data
- Variable used in the test execution flow of the Application Test definition
- User function of the Classic Test definition
- Analysis function of the Classic Test definition

The following characters can be used for the variable name.

- Alphabetic characters A to Z and a to z, case sensitive
- \_ (underscore)
- Numeric characters 0 to 9
- @ (at mark)

Numeric characters and at mark cannot be used for the first character of variable name.

---

### NOTE

#### About variable names

The variable name must be unique. If a variable is defined as a local variable in an application test definition and its name is the same as one of the following variables, the local variable is effective and the following variable is not effective.

- variable that defines a built-in function in a test setup
- variable that defines a read-out function in a test setup
- variable defined as a test parameter
- variable defined as a device parameter
- variable defined as an analysis parameter

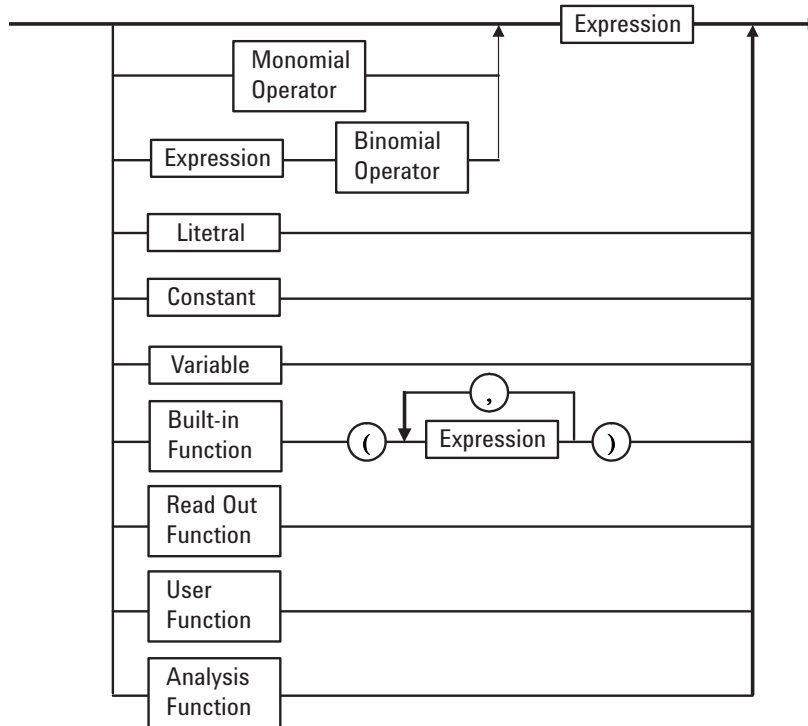
The local variables are defined by using the Local Variable Definition statement in the Test Contents setup displayed on the Test Definition window. The test parameters and the device parameters are defined in the Test Specification setup, and the analysis parameters are defined in the Test Output setup.

## Expressions

Figure 5-1 shows the expression effective for variable definition. The elements are explained below.

Figure 5-1

### Syntax of Expression



## Constants

The following scientific constants are available.

- q: electric charge.  $1.602177 \times 10^{-19}$
- k: Boltsmann's constant.  $1.380658 \times 10^{-23}$
- e: space permittivity.  $8.854188 \times 10^{-12}$

## Literals

Numeric type and string type are available.

- Numeric type

To enter a numeric value, type as follows, where [ ] is optional.

`Dd[.d][E[S]d][s]`

D: Numeric character 0 to 9

d: This means repeating D 0 times or more.

E: Exponent symbol, E or e

S: Sign, + or -

s: Prefix symbol. The subsequent symbols (y, z, a, f, p, n, u, m, k, M, G) separated by the comma mean the value following the colon.

y:  $10^{-24}$ , z:  $10^{-21}$ , a:  $10^{-18}$ , f:  $10^{-15}$ , p:  $10^{-12}$ , n:  $10^{-9}$ , u:  $10^{-6}$ , m:  $10^{-3}$ ,  
k:  $10^3$ , M:  $10^6$ , G:  $10^9$

- String type

Enclose the string by using the double quotes. To contain the double quotes into the string, type as shown below. The example string is ABC"DEF.

```
"ABC " "DEF "
```

## Monomial Operators and Binomial Operators

Table 5-1 shows monomial operators and binomial operators available for an expression. The table also shows the priority of operators and the type of operand. For the expression that contains the same priority operators, the operations are performed from left to right. Module type variable is one of SMU, HPSMU, HCSMU, DHCSMU, HVSMU, MPSMU, MCSMU, HRSMU, HRSMU/ASU, CMH, CML, UHCU500, UHCU1500, UHVU, or HVMCU.

---

### NOTE

#### Operation between arrays

Operation is performed between data at the same index.

---

**Table 5-1 Monomial Operator and Binomial Operator**

Priority	Operator	Type of operand			
		Numeric	Vector <sup>a</sup>	String	Module
High	()	Yes	Yes	Yes	Yes
	NOT	Yes	Yes	No	No
	^	Yes	Yes	No	No
	* or /	Yes	Yes	No	No
	+ or -, monomial	Yes	Yes	+: Yes, -: No	No
	+ or -, binomial	Yes	Yes	+: Yes, -: No	No
	>, >=, <, <=	Yes	Yes	Yes	No
	=, <>	Yes	Yes	Yes	No
Low	AND	Yes	Yes	No	No
	XOR	Yes	Yes	No	No
	OR	Yes	Yes	No	No

- a. Two-dimensional array. The primary plane is for the primary sweep index, and the secondary plane is for the secondary sweep index.

### Built-in Functions

Agilent EasyEXPERT built-in function can be used. See “Built-in Functions” on page 5-7.

### Read-out Functions

Agilent EasyEXPERT built-in read-out function can be used. See “Read-out Functions” on page 5-16.

### User Functions and Analysis Functions

User functions and analysis functions are the user-defined functions that consist of the variable name, unit, and expression. The functions are effective for numeric or vector type data. To define the functions, see “Function Setup” on page 2-42.

Numeric literal, scientific constants (q, k, e), variable to store the test result data, and a built-in function that returns the numeric or vector type data can be used in the expression. The read-out function can also be used for the analysis functions.

---

## Built-in Functions

This section describes Agilent EasyEXPERT built-in functions. The functions appear in alphabetical order. In the function name, the character in parentheses is a variable.

### **abs(A)**

Returns the absolute value of the expression A.

**Example** `B=abs(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

### **acos(A)**

Returns the arccosine value of the expression A. The A value must be between -1 and 1.

**Example** `B=acos(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

### **asin(A)**

Returns the arcsine value of the expression A. The A value must be between -1 and 1.

**Example** `B=asin(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

### **at(A,B)**

Finds a data element in the expression A, and returns the value. The data element is specified by index B. The index number starts from 1. If B is not an integer, a linear interpolated A value will be returned.

**Example** `C=at(A,B)`

**Data type** A: vector or numeric, B: numeric, C: vector.

### **at(A,B,C)**

Finds a data element in the expression A, and returns the value. The data element is specified by the indices B and C. B is the index for the primary plane of a two-dimensional array and C is the index for the secondary plane. The index number starts from 1.

If B is not an integer, a linear interpolated A value will be returned. The linear interpolation will be executed between A[n,C] and A[n+1,C], where  $n < B < n+1$ .

If C is not an integer, C will be n, where  $n < C < n+1$ .

**Example** `D=at(A,B,C)`

**Data type** A: vector or numeric, B: numeric, C: numeric, D: numeric.

## Built-in Programming Tool

### Built-in Functions

- atan(A)** Returns the arctangent value of the expression A.
- Example**  $B = \text{atan}(A)$
- Data type** A: numeric, B: numeric. Or A: vector, B: vector.
- atan2(A,B)** Returns the radian value of the angle made by the X-axis and the line that passes through the coordinates (0, 0) and (A, B).
- Example**  $C = \text{atan2}(A, B)$
- Data type** A: numeric, B: numeric, C: numeric. If A and/or B is vector, C: vector.
- avg(A)** Returns the average value of data in the expression A. For subordinate sweep measurement, this function returns the average values for each primary sweep.
- Example**  $B = \text{avg}(A)$
- Data type** A: vector or numeric, B: vector.
- ceil(A)** Returns the minimum integer value more than or equal to the expression A.
- Example**  $B = \text{ceil}(A)$
- Data type** A: numeric, B: numeric. Or A: vector, B: vector.
- cond(A,B,C,D)** This function does the following.
- If  $A < B$ , returns C.
  - If  $A \geq B$ , returns D.
- If the A or B value is invalid, the value for the previous measurement index number is used for the comparison.
- Example**  $E = \text{cond}(A, B, C, D)$
- Data type** A, B, C, D: numeric, E: numeric. If A, B, C, and/or D is vector, E: vector.
- cos(A)** Returns the cosine value of the expression A.
- Example**  $B = \text{cos}(A)$
- Data type** A: numeric, B: numeric. Or A: vector, B: vector.



**delta(A)**

Returns the difference of the expression A. Calculation is performed as follows.

$$D_n = (A_2 - A_1) \quad \text{when } n = 1$$

$$D_n = (A_{n+1} - A_{n-1}) / 2 \quad \text{when } 1 < n < N$$

$$D_n = (A_N - A_{N-1}) \quad \text{when } n = N$$

where,

- D<sub>n</sub>**            difference for the index number n
- A<sub>n</sub>**            A value for the index number n
- N**                number of sweep steps or number of samples

For subordinate sweep, the first index number of each primary sweep is always 1.

If A is a data variable for a secondary sweep source, this function returns the sweep step value of the secondary sweep.

For frequency data of C-V sweep, the calculation is performed for all frequencies.

**Example**    `B=delta(A)`

**Data type**    A: vector or numeric, B: vector.

**diff(A,B)**

Returns the differential coefficient of the expression A by B. Calculation is performed as follows.

$$A'_n = (A_2 - A_1) / (B_2 - B_1) \quad \text{when } n = 1$$

$$A'_n = (A_{n+1} - A_{n-1}) / (B_{n+1} - B_{n-1}) \quad \text{when } 1 < n < N$$

$$A'_n = (A_N - A_{N-1}) / (B_N - B_{N-1}) \quad \text{when } n = N$$

where,

- A'<sub>n</sub>**:            differential coefficient for the index number n
- A<sub>n</sub>**:            A value for the index number n
- B<sub>n</sub>**:            B value for the index number n
- N**:                number of sweep steps or number of samples

For subordinate sweep, the first index number of each primary sweep is always 1.

**Example**    `C=diff(A,B)`

**Data type**    A, B: vector or numeric, C: vector.

## Built-in Programming Tool

### Built-in Functions

- dim1Size(A)** Returns the number of elements in the primary plane of the vector variable A.  
**Example** `B=dim1Size(A)`  
**Data type** A: vector, B: numeric.
- dim2Size(A)** Returns the number of elements in the secondary plane of the vector variable A.  
**Example** `B=dim2Size(A)`  
**Data type** A: vector, B: numeric.
- exp(A)** Raises e (exponential) to the power of the expression A. The following example enters  $e^A$  calculation result to the variable B.  
**Example** `B=exp(A)`  
**Data type** A: numeric, B: numeric. Or A: vector, B: vector.
- floor(A)** Returns the maximum integer value less than or equal to the expression A.  
**Example** `B=floor(A)`  
**Data type** A: numeric, B: numeric. Or A: vector, B: vector.
- getNumericData(A)** Returns data of the numeric variable specified by A. This function is effective for the variables defined in the Function Setup or Display Setup of the last Classic Test or the analysis parameters of the last Application Test. The variables are valid in a test definition.  
**Example** `B=getNumericData("A")`  
**Data type** A: string (name of numeric variable), B: numeric.
- getVectorData(A)** Returns data of the vector variable specified by A. This function is effective for the variables defined in the Function Setup or Display Setup of the last Classic Test or the analysis parameters of the last Application Test. The variables are valid in a test definition.  
**Example** `B=getVectorData("A")`  
**Data type** A: string (name of vector variable), B: vector.
- index(A,B)** Finds the string B in the string A, and returns the index number where the string B is first found. The index 0 indicates the first character. The following example returns 3.  
**Example** `C=index("AGILENT", "L")`  
**Data type** C: numeric.

**integ(A,B)**

Performs numerical integration of the expression A by B. Calculation is performed as follows.

When  $n = 1$ ,  $\alpha_n = 0$

When  $n > 1$ ,  $\alpha_n$  is defined as follows:

$$\alpha_n = \frac{1}{2} \sum_{i=2}^n (y_i + y_{i-1})(x_i - x_{i-1})$$

where,

$\alpha_n$ : integral of A for the index number n

$y_i$ : A value for the index number i

$x_i$ : B value for the index number i

If A or B contain some invalid values, the invalid values are ignored for the calculation.

For subordinate sweep, the first index number of each primary sweep is always 1.

**Example** `C=integ(A,B)`

**Data type** A, B: vector or numeric, C: vector.

**isValid(A)**

Returns 1 if the expression A is valid and is not double.NaN, or else returns 0.

**Example** `B=isValid(A)`

**Data type** A: numeric, B: numeric.

**lgt(A)**

Returns the logarithm (base 10) of the expression A.

If  $A=0$ , -Overflow is returned with the status "Arithmetic error".

If A is a negative value, the logarithm of the absolute value is returned with the status "Arithmetic error".

**Example** `B=lgt(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

## Built-in Programming Tool

### Built-in Functions

#### **log(A)**

Returns the logarithm (base e) of the expression A.

If A=0, -Overflow is returned with the status “Arithmetic error”.

If A is a negative value, the logarithm of the absolute value is returned with the status “Arithmetic error”.

**Example**  $B = \log(A)$

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

#### **mavg(A,B)**

Returns the moving average value of data in the expression A. B specifies how many data to use for the average. The moving average at the index number n is defined as follows.

When  $n \leq r$

$$\bar{x}_n = \frac{1}{r+n} \sum_{i=1}^{n+r} x_i$$

When  $r < n \leq N-r$

$$\bar{x}_n = \frac{1}{2r+1} \sum_{i=n-r}^{n+r} x_i$$

When  $N-r < n$

$$\bar{x}_n = \frac{1}{r+N-n+1} \sum_{i=n-r}^N x_i$$

where,

$x_n$  : moving average of A for the index number n

$x_i$  : A value for the index number i

$r$  : B value

$N$  : number of sweep steps or number of samples

If A contains some invalid values, the invalid values are ignored for the calculation.

For subordinate sweep, the first index number of each primary sweep is always 1.

**Example**  $C = \text{mavg}(A, B)$

**Data type** A: vector or numeric, B: numeric, C: vector.

**max(A)** Returns the maximum value of data in the expression A. For subordinate sweep measurement, this function returns the maximum value of the primary sweep for each secondary sweep step.

If there are invalid values in A, invalid values are ignored.

**Example** `B=max(A)`

**Data type** A: vector or numeric, B: vector.

**min(A)** Returns the minimum value of data in the expression A. For subordinate sweep measurement, this function returns the minimum value of the primary sweep for each secondary sweep step.

If A contains invalid values, the invalid values are ignored.

**Example** `B=min(A)`

**Data type** A: vector or numeric, B: vector.

**readThermometerOfN1265A(A)** Performs the temperature measurement by using the thermometer channel specified by the expression A, and returns the measured value. If this function is executed when a classic test is in progress, this function returns the last measured temperature value. To enable the thermometer channel, see “UHC Expander / Fixture” on page 1-72. If the thermometer channel is disabled, this function returns NaN.

**Example** `B=readThermometerOfN1265A(A)`

**Data type** A: numeric (1 or 2), B: numeric (°C).

**setDeviceId(A)** Sets the value of the string variable A to the Device ID parameter of the test result record. This value is effective for the test result records created after this function call. If the A value is an empty string, this function does nothing.

**Example** `B=setDeviceId(A)`

**Data type** A: string, B: numeric. B (returned value) is always 0.

**sin(A)** Returns the sine value of the expression A.

**Example** `B=sin(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

**sqrt(A)** Returns the square root of the expression A.

**Example** `B=sqrt(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

## Built-in Programming Tool

### Built-in Functions

**storeAt(A,B,C,D)** Finds a data element in the expression A, and replaces it with the D value, and returns the result vector data. The data element is specified by the indices B and C. B is the index for the primary plane of a two-dimensional array and C is the index for the secondary plane. The index number starts from 1. The following example replaces the A[5, 5] value with the D value and enters the result vector data to the variable E.

**Example** `E=storeAt(A,5,5,D)`

**Data type** A: vector or numeric, B, C, D: numeric, E: vector.

**string(A)** Translates the expression A to a string data, and returns it.

**Example** `B=string(A)`

**Data type** A: numeric or module, B: string.

---

#### NOTE

#### Translating the variable defined with the numeric-symbol assignment

If the argument of the *string* function is a numeric variable defined with the numeric-symbol assignment, the *string* function execution result will be the symbol assigned to the numeric value, not the string value just translated from the numeric value. The numeric-symbol assignment is defined in the “Test Specification” on page 3-5.

For the following example setup of the numeric-symbol assignment, `string(Mode)` execution result at `Mode=0` is the string OFF, not the string 0.

Parameter Mode, Value=0, Symbol=OFF

---

**substring(A,B,C)** Extracts a string from the string A, and returns it. B is the index used to specify the first character to extract, and C is the number of characters to extract. The index number starts from 0. If B or C is not an integer, digits over the decimal point are used as the integer. The following example returns G.

**Example** `D=substring("AGILENT",1,1)`

**Data type** A: string, B, C: numeric, D: string.

**swmToModule(A)** Searches the measurement module connected to Agilent B2200/E5250 switching matrix input port number A. And returns the module name.

**Example** `B=swmToModule(A)`

**Data type** A: string (Input port number 1 to 14), B: module.

**tan(A)** Returns the tangent value of the expression A.

**Example** `B=tan(A)`

**Data type** A: numeric, B: numeric. Or A: vector, B: vector.

**value(A)**

Translates the string A to a numeric data, and returns it. If the A value is invalid as numeric literal, 0 is returned.

**Example** `B=value(A)`

**Data type** A: string, B: numeric.

## Read-out Functions

The read-out functions are built-in functions for reading various values related to a analysis tool which is marker, cursor, or line. You can use these functions to perform complex analysis of the measurement results.

To read the value of the read-out function, use a numeric variable. The following example reads the X intercept of the line 1 and enters it to the variable A.

**Example**    `A=@L1X`

---

### NOTE

In this section, *n* of the function name is an integer (1 to 8), and *m* is 1 or 2.

You cannot assign a data variable that includes a read-out function to an axis of a graphics plot.

If the analysis tool that is referred to by the read-out function is not displayed, the read-out function uses the position at which it was most recently displayed. If the analysis tool has not been displayed, the read-out function returns invalid data.

In the application test definition, to use a read-out function in a component such as ASSIGN, IF, FOR, Application Test, and Classic Test, place the Auto Analysis component before the component, and set the analysis tool associated with the read-out function.

---

## Functions for Marker

### @MI

Returns the index number of measurement data at the marker location.

If interpolation mode is enabled and the marker is located between the measurement data, this function returns a non-integer value.

### @MX

Returns the value of the X coordinate at the marker location.

### @MY

Returns the value of the Y coordinate at the marker location.

If there are two Y axes, this function returns the value for the selected axis.

### @MY $n$

Returns the value of the Y $n$  coordinate at the marker location ( $n$ : 1 to 8, integer).



## Functions for Cursor

- @CX** Returns the value of the X coordinate at the active cursor position.
- @CY** Returns the value of the Y coordinate at the active cursor position. If there are two Y axes, this function returns the value for the selected axis.
- @CY $n$**  Returns the value of the Y $n$  coordinate at the active cursor position ( $n$ : 1 to 8, integer).

## Functions for Line

- @LmCO** Returns the correlation coefficient of the regression for Line  $m$  ( $m$ : 1 or 2). Line  $m$  must be in regression mode. If not, this function returns invalid data.

- @LmG** Returns the slope of Line  $m$  ( $m$ : 1 or 2). If there are two Y axes, this function returns the value for the selected axis. This function calculates the slope by using the following formula:

- If the X and Y axes are both linear scale:  
$$\alpha = (y_1 - y_0) / (x_1 - x_0)$$
- If the X axis is log scale, and the Y axis is linear scale:  
$$\alpha = (y_1 - y_0) / (\log x_1 - \log x_0)$$
- If the X axis is linear scale, and the Y axis is log scale:  
$$\alpha = (\log y_1 - \log y_0) / (x_1 - x_0)$$
- If the X and Y axes are both log scale:  
$$\alpha = (\log y_1 - \log y_0) / (\log x_1 - \log x_0)$$

where,

$\alpha$  : Slope of Line  $m$

$x_0, y_0, x_1, y_1$  : X and Y coordinate values at the two points where Line  $m$  intercepts the perimeter of the plotting area.

- @LmG $n$**  Returns the slope of Line  $m$  ( $m$ : 1 or 2) for the Y $n$  axis ( $n$ : 1 to 8, integer). This function calculates the slope by using the following formula:

## Built-in Programming Tool Read-out Functions

- If the X and Y<sub>n</sub> axes are both linear scale:  
$$\alpha = (y_1 - y_0) / (x_1 - x_0)$$
- If the X axis is log scale, and the Y<sub>n</sub> axis is linear scale:  
$$\alpha = (y_1 - y_0) / (\log x_1 - \log x_0)$$
- If the X axis is linear scale, and the Y<sub>n</sub> axis is log scale:  
$$\alpha = (\log y_1 - \log y_0) / (x_1 - x_0)$$
- If the X and Y<sub>n</sub> axes are both log scale:  
$$\alpha = (\log y_1 - \log y_0) / (\log x_1 - \log x_0)$$

where,

$\alpha$  : Slope of Line *m*

$x_0, y_0, x_1, y_1$  : X and Y coordinate values at the two points where Line *m* intercepts the perimeter of the plotting area.

**@LmX** Returns the X intercept value (Y=0) of Line *m* (*m*: 1 or 2).  
If Line *m* is horizontal, this function returns invalid data.

**@LmY** Returns the Y intercept value (X=0) of Line *m* (*m*: 1 or 2).  
If there are two Y axes, this function returns the value for the selected axis.  
If Line *m* is vertical, this function returns invalid data.

**@LmY<sub>n</sub>** Returns the Y<sub>n</sub> (*n*: 1 to 8) intercept value (X=0) of Line *m* (*m*: 1 or 2).  
If Line *m* is vertical, this function returns invalid data.

## Functions for Two Line Display

**@IX** Returns the value of the X coordinate at the intersection of Line 1 and Line 2.  
This function calculates the intersection by using the following formula.

$$X = (y_2 - y_1) / (\alpha_2 - \alpha_1)$$

where,

$X$  : Value of the X coordinate at the intersection. If the X axis is log scale, this function returns  $10^X$ .

$y_m$  : Y intercept value of Line  $m$  ( $m$ : 1 or 2). If the Y axis is log scale,  $y_m$  is the log value of the Y intercept of Line  $m$ .

$\alpha_m$  : Slope of Line  $m$

If Line 1 is parallel to Line 2, this function returns invalid data with the status “Arithmetic error”.

### @IY

Returns the value of the Y coordinate at the intersection of Line 1 and Line 2.

If there are two Y axes, this function returns the value for the selected axis.

This function calculates the intersection by using the following formula:

$$Y = y_1 + \alpha_1 \times (y_2 - y_1) / (\alpha_1 - \alpha_2)$$

where,

$Y$  : Value of the Y coordinate at the intersection. If the Y axis is log scale, this function returns  $10^Y$ .

$y_m$  : Y intercept value of Line  $m$  ( $m$ : 1 or 2). If the Y axis is log scale,  $y_m$  is the log value of the Y intercept of Line  $m$ .

$\alpha_m$  : Slope of Line  $m$

If Line 1 is parallel to Line 2, this function returns invalid data with the status “Arithmetic error”.

### @IYn

Returns the value of the  $Y_n$  ( $n$ : 1 to 8, integer) coordinate at the intersection of Line 1 and Line 2.

This function calculates the intersection by using the following formula:

$$Y_n = y_1 + \alpha_1 \times (y_2 - y_1) / (\alpha_1 - \alpha_2)$$

where,

$Y_n$  : Value of the  $Y_n$  coordinate at the intersection. If the  $Y_n$  axis is log scale, this function returns  $10^{Y_n}$ .

$y_m$  :  $Y_n$  intercept of Line  $m$  ( $m$ : 1 or 2). If the  $Y_n$  axis is log scale,  $y_m$  is the log value of the  $Y_n$  intercept of Line  $m$ .

$\alpha_m$  : Slope of Line  $m$

If Line 1 is parallel to Line 2, this function returns invalid data with the status “Arithmetic error”.

## Script Program Statements

The program statements are used to define the test execution flow on the Test Contents tab screen of the Test Definition window. On the screen, click the Program Component tab. The program statements are listed in the component selection area. See also “Program Component” on page 3-26.

### Local Variable Definition

This statement defines the local variables used in the test definition. For the setup editor, see “Local Variable Definition” on page 3-24. For the available names and variable types, see “Variables and Expressions” on page 5-3.

The local variables are effective only for this test definition.

### ASSIGN

This statement enters a value or expression to the variable as follows:

<variable> = <definition>

where, <variable> is the variable to define, and <definition> is the value or expression.

### BLOCK

Beginning of the BLOCK/END BLOCK block. The statements are added to the program list as a pair. The test execution flow (program) must be defined between the statements. Optionally, the following statements can be used in this block.

- ON ERROR

This statement enables an event-initiated branch performed when an error occurs. The routine to be performed when error occurs must be defined between ON ERROR and END BLOCK or between ON ERROR and FINALLY.

- FINALLY

This statement enables an event-initiated branch performed just before terminating the block. The routine to be performed before terminating the block must be defined between FINALLY and END BLOCK.

The statements or components between the BLOCK and END BLOCK statements are performed from top-down. If no error occurs during program execution, the routine below FINALLY is executed and the block is terminated.

If an error occurs, the error code is stored in the @ERROR system variable, the routine below ON ERROR is executed, and the block is terminated. If FINALLY is used, the routine below FINALLY is executed before the block is terminated.

<b>COMMENT</b>	This statement enters a comment in the program line.
<b>END</b>	This statement terminates program execution. After the END statement, the routines defined by the FINALLY statement are performed.
<b>ERROR</b>	This statement causes an error. However, no error code is stored in the @ERROR system variable because of the fictitious error.
<b>FOR</b>	<p>Beginning of the FOR/NEXT loop block. FOR and NEXT are added to the program list as a pair. The statements repeatedly performed when looping must be defined between the statements. The FOR/NEXT statements are entered as follows:</p> <pre>FOR &lt;variable&gt; = &lt;initial value&gt; TO &lt;terminate value&gt; STEP &lt;step value&gt; NEXT &lt;variable&gt;</pre> <p>where, &lt;variable&gt; is a variable used for the loop counter. &lt;initial value&gt;, &lt;terminate value&gt;, and &lt;step value&gt; are values for the loop counter start value, stop value, and step value respectively.</p> <p>The EXIT FOR statement is available to exit from the loop before the loop count reaches &lt;terminate value&gt;. After looping, the next line of NEXT is performed.</p>
<b>IF</b>	<p>Beginning of the IF/END IF block. The statements are added to the program list as a pair. The IF statement specifies the primary branching condition. The routines performed when the condition is satisfied are defined between the statements. The statements are entered as follows:</p> <pre>IF &lt;expression&gt; END IF</pre> <p>where, &lt;expression&gt; is the condition of branching. Optionally the following statements can be used in this block.</p> <ul style="list-style-type: none"><li>• ELSE IF &lt;expression&gt; This statement specifies the additional branching condition used when the previous condition is not satisfied, and defines the routines performed when this branching condition is satisfied.</li><li>• ELSE This statement defines the routines performed when no branching condition is satisfied.</li></ul>

## Built-in Programming Tool

### Script Program Statements

#### LOOP

Beginning of the LOOP/END LOOP block. LOOP and END LOOP are added to the program list as a pair. The statements repeatedly performed during looping should be defined between the statements.

The EXIT LOOP statement is available to exit from the loop. After looping, the next line of END LOOP is performed.

#### Example 1

Test definition name: Vth\_gmMax

Local Variables Definition

BLOCK

```
!***** Integration Time Setting *****
```

```
IF IntegTime="SHORT"
```

```
    IntegMode = "MANUAL"
```

```
    AdcFactor = 8
```

```
ELSE IF IntegTime="MEDIUM"
```

```
    IntegMode = "PLC"
```

```
    AdcFactor = 1
```

```
ELSE IF IntegTime="LONG"
```

```
    IntegMode = "PLC"
```

```
    AdcFactor = 16
```

```
END IF
```

```
!***** I-V Sweep *****
```

```
Vth_gmmax
```

```
!***** Analysis *****
```

```
Auto Analysis
```

```
END BLOCK
```

Line 1: Defines local variables.

Line 2: Beginning of program. Program ends by the END BLOCK statement.

Line 3: Comment line. The COMMENT statement is used.

Lines 4 to 13: Changes the integration time setting by the variable IntegTime value.

Line 14: Comment line. The COMMENT statement is used.

Line 15: Executes Vth\_gmmax of the Application Test component to perform Id-Vg measurement.

Line 16: Comment line. The COMMENT statement is used.

Line 17: Executes Auto Analysis of the Analysis component.

## Example 2

Test definition name: NandFlash IV-Write-IV

Local Variables Definition

BLOCK

```
!***** Integration Time Setting *****
```

```
IF IntegTime="SHORT"
```

```
    IntegMode = "MANUAL"
```

```
    AdcFactor = 8
```

```
ELSE IF IntegTime="MEDIUM"
```

```
    IntegMode = "PLC"
```

```
    AdcFactor = 1
```

```
ELSE IF IntegTime="LONG"
```

```
    IntegMode = "PLC"
```

```
    AdcFactor = 16
```

```
END IF
```

```
!***** I/V Sweep & Analysis Initial Characteristics *****
```

```
ForcePG2
```

```
Id-Vg_Initial
```

```
ResetPG
```

```
Auto Analysis
```

(continues to the next page)

Line 1: Defines local variables.

Line 2: Beginning of program. Program ends by the END BLOCK statement.

Line 3: Comment line. The COMMENT statement is used.

Lines 4 to 13: Changes the integration time setting by the variable IntegTime value.

Line 14: Comment line. The COMMENT statement is used.

Line 15: Executes ForcePG2 of the Application Test component to apply pulse from the Output 2 terminal of Agilent 81110A pulse generator connected to the B1500.

Line 16: Executes Id-Vg\_Initial of the Application Test component to perform Id-Vg measurement.

Line 17: Executes ResetPG of the Application Test component to reset Agilent 81110A.

Line 18: Executes Auto Analysis of the Analysis component.

## Built-in Programming Tool

### Script Program Statements

(continues from the previous page)

```
VthInitial = getNumericData("VthBefore")
VgateList = getVectorData("Vgate")
IdInitialList = getVectorData("Idrain")
!***** Force Pulse *****
ForcePG1
ResetPG
!***** I/V Sweep & Analysis after Write *****
ForcePG2
Id-Vg_Written
ResetPG
Auto Analysis
VthWritten = getNumericData("VthAfter")
IdWrittenList = getVectorData("Idrain")
END BLOCK
```

Lines 19 to 21: Enters the value of the variables (VthBefore, Vgate, Idrain) defined in the Function Setup or Display Setup of the Id-Vg\_Initial test into the local variables (VthInitial, VgateList, IdInitialList). The ASSIGN statements are used.

Line 22: Comment line. The COMMENT statement is used.

Line 23: Executes ForcePG1 of the Application Test component to perform the NAND-type flash memory write operation.

Line 24: Executes ResetPG of the Application Test component.

Line 25: Comment line. The COMMENT statement is used.

Line 26: Executes ForcePG2 of the Application Test component.

Line 27: Executes Id-Vg\_Written of the Application Test component to perform Id-Vg measurement.

Line 28: Executes ResetPG of the Application Test component.

Line 29: Executes Auto Analysis of the Analysis component.

Lines 30 to 31: Enters the value of the variables (VthAfter, Idrain) defined in the Function Setup or Display Setup of the Id-Vg\_Written test into the local variables (VthWritten, IdWrittenList). The ASSIGN statements are used.

Line 32: End of program