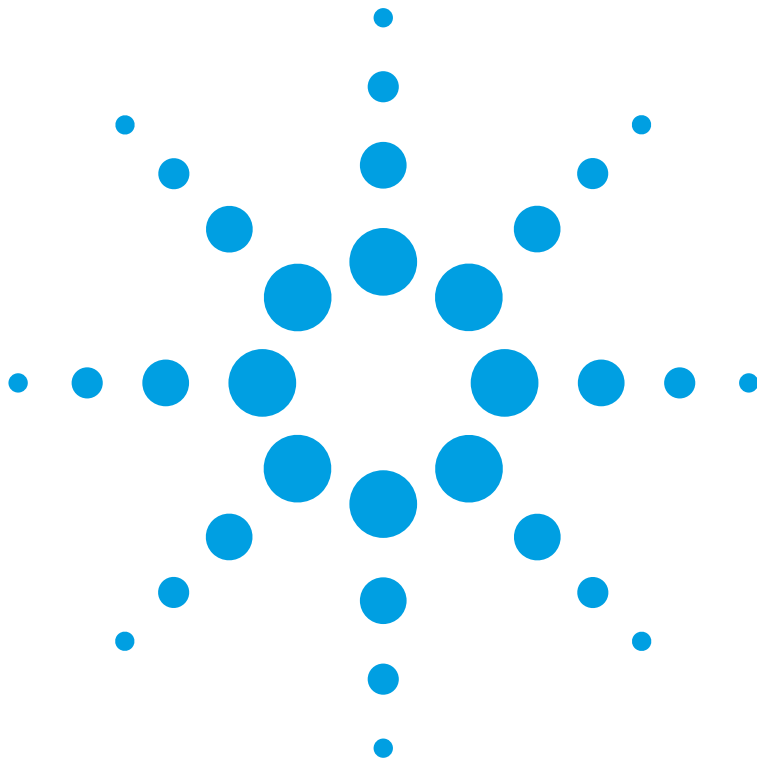


**Agilent 8703B Lightwave
Component Analyzer
User's Guide**



Agilent Technologies

© Copyright
Agilent Technologies 2001
All Rights Reserved. Reproduction, adaptation, or translation without prior written permission is prohibited, except as allowed under copyright laws.

Agilent Part No. 08703-90201
Printed in USA
July 2001

Agilent Technologies
Lightwave Division
3910 Brickway Boulevard Santa
Rosa, CA 95403, USA

Notice.

The information contained in this document is subject to change without notice. Companies, names, and data used in examples herein are fictitious unless otherwise noted. Agilent Technologies makes no warranty of any kind with regard to this material, including but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Restricted Rights Legend.

Use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 for DOD agencies, and subparagraphs (c) (1) and (c) (2) of the Commercial Computer Software Restricted Rights clause at FAR 52.227-19 for other agencies.

Warranty.

This Agilent Technologies instrument product is warranted against defects in material and workmanship for a period of one year from date of shipment. During the warranty period, Agilent Technologies will, at its option, either repair or replace products which prove to be defective. For warranty service or repair, this product must be returned to a service facility designated by Agilent Technologies. Buyer shall prepay shipping charges to Agilent Technologies and Agilent Technologies shall pay shipping charges to return the product to Buyer. However, Buyer shall pay all shipping charges, duties, and taxes for products returned to Agilent Technologies from another country.

Agilent Technologies warrants that its software and firmware designated by Agilent Technologies for use with an instrument will execute its programming instructions when properly installed on that instrument. Agilent Technologies does not warrant that the operation of the instrument, or software, or firmware will be uninterrupted or error-free.

Limitation of Warranty.

The foregoing warranty shall not apply to defects resulting from improper or inadequate maintenance by Buyer, Buyer-supplied software or interfacing, unauthorized modification or misuse, operation outside of the environmental specifications for the product, or improper site preparation or maintenance.

No other warranty is expressed or implied. Agilent Technologies specifically disclaims the implied warranties of merchantability and fitness for a particular purpose.

Exclusive Remedies.

The remedies provided herein are buyer's sole and exclusive remedies. Agilent Technologies shall not be liable for any direct, indirect, special, incidental, or consequential damages, whether based on contract, tort, or any other legal theory.

Safety Symbols. CAUTION

The *caution* sign denotes a hazard. It calls attention to a procedure which, if not correctly performed or adhered to, could result in damage to or destruction of the product. Do not proceed beyond a caution sign until the indicated conditions are fully understood and met.

WARNING

The *warning* sign denotes a hazard. It calls attention to a procedure which, if not correctly performed or adhered to, could result in injury or loss of life. Do not proceed beyond a warning sign until the indicated conditions are fully understood and met.



The instruction manual symbol. The product is marked with this warning symbol when it is necessary for the user to refer to the instructions in the manual.



The laser radiation symbol. This warning symbol is marked on products which have a laser output.



The AC symbol is used to indicate the required nature of the line module input power.



The ON symbols are used to mark the positions of the instrument power line switch.



The OFF symbols are used to mark the positions of the instrument power line switch.



The CE mark is a registered trademark of the European Community.



The CSA mark is a registered trademark of the Canadian Standards Association.



The C-Tick mark is a registered trademark of the Australian Spectrum Management Agency.

ISM1-A

This text denotes the instrument is an Industrial Scientific and Medical Group 1 Class A product.

Typographical Conventions.

The following conventions are used in this book:

Key type for keys or text located on the keyboard or instrument.

Softkey type for key names that are displayed on the instrument's screen.

Display type for words or characters displayed on the computer's screen or instrument's display.

User type for words or characters that you type or enter.

Emphasis type for words or characters that emphasize some point or that are used as place holders for text that you type.

1. Connector Care & ESD Information

- Cleaning Connections for Accurate Measurements 1-2
- Caring for Electrical Connections 1-11
- Electrostatic Discharge Information 1-12

2. Making Measurements

- Making Measurements 2-2
- Making a Basic Measurement 2-3
- Using Display Functions 2-4
- Using Markers 2-17
- Using the List Mode to Test a Device 2-37
- Using Limit Lines to Test a Device 2-42
- Using Ripple Limits to Test a Device 2-50
- Using Bandwidth Limits to Test a Bandpass Filter 2-60
- Using Test Sequencing 2-66
- Using Test Sequencing to Test a Device 2-77
- Magnitude and Phase Comparisons of Lightwave Receivers 2-83
- Mathematically Combining Device Responses 2-91
- Using a Coefficient Model for Device Response and Design Model Comparison 2-95
- Using an External Laser 2-98

3. Optimizing Measurement Results

- Optimizing Measurement Results 3-2
- Taking Care of Connectors 3-3
- Increasing Measurement Accuracy 3-4
- Increasing Dynamic Range 3-7
- Reducing Noise 3-8
- Reducing Receiver Crosstalk 3-9

4. Calibrating for Increased Measurement Accuracy

- Calibrating for Increased Measurement Accuracy 4-2
- Calibration Considerations 4-3
- Procedures for Error Correcting Your Measurements 4-9
- O/O Response Calibration 4-11
- O/O Response and Isolation Calibration 4-13
- E/O Response Calibration 4-16
- E/O Reflection Sensitivity Calibration 4-17
- E/O Response and Isolation Calibration 4-18
- E/O Response and Match Calibration 4-20
- O/E Response Calibration 4-23
- O/E Response and Isolation Calibration 4-24
- O/E Response and Match Calibration 4-26
- Modifying Lightwave Calibration Kits 4-29
- Modifying an Electrical Calibration Kit 4-30
- Verify Performance 4-39

5. Verifying Measurement Accuracy

- Verifying Measurement Accuracy 5-2
- Agilent Technologies Service Offices 5-8

6. Saving and Recalling Measurements and Data

- Saving and Recalling Measurements and Data 6-2
- Saving and Recalling Instrument States 6-3
- Saving an Instrument State 6-5
- Saving Measurement Results 6-6
- Re-Saving an Instrument State 6-18
- Deleting a File 6-19
- Renaming a File 6-20
- Recalling a File 6-21
- Formatting a Disk 6-22
- Solving Problems with Saving or Recalling Files 6-23

7. Printing Measurement Results

- Printing Measurements Results 7-2
- Printing or Plotting Your Measurement Results 7-3
- Configuring a Print Function 7-4
- Defining a Print Function 7-6
- Printing One Measurement Per Page 7-8
- Printing Multiple Measurements Per Page 7-9
- Configuring a Plot Function 7-10
- Defining a Plot Function 7-15
- Plotting One Measurement Per Page Using a Pen Plotter 7-19
- Plotting Multiple Measurements Per Page Using a Pen Plotter 7-20
- To View Plot Files on a PC 7-22
- Outputting Plot Files from a PC to a Plotter 7-23
- Outputting Plot Files from a PC to an HPGL Compatible Printer 7-24
- Outputting Single Page Plots Using a Printer 7-26
- Outputting Multiple Plots to a Single Page Using a Printer 7-27
- Plotting Multiple Measurements Per Page from Disk 7-28
- Titling the Displayed Measurement 7-32
- Configuring the Analyzer to Produce a Time Stamp 7-33
- Aborting a Print or Plot Process 7-34
- Printing or Plotting the List Values or Operating Parameters 7-35
- Solving Problems with Printing or Plotting 7-36

1

Cleaning Connections for Accurate Measurements	1-2
Choosing the Right Connector	1-2
Inspecting Connectors	1-4
Cleaning Connectors	1-8
Caring for Electrical Connections	1-11
Electrostatic Discharge Information	1-12
Reducing ESD Damage	1-13

Connector Care & ESD Information

Cleaning Connections for Accurate Measurements

Today, advances in measurement capabilities make connectors and connection techniques more important than ever. Damage to the connectors on calibration and verification devices, test ports, cables, and other devices can degrade measurement accuracy and damage instruments. Replacing a damaged connector can cost thousands of dollars, not to mention lost time! This expense can be avoided by observing the simple precautions presented in this document. This document also contains a brief list of tips for caring for electrical connectors.

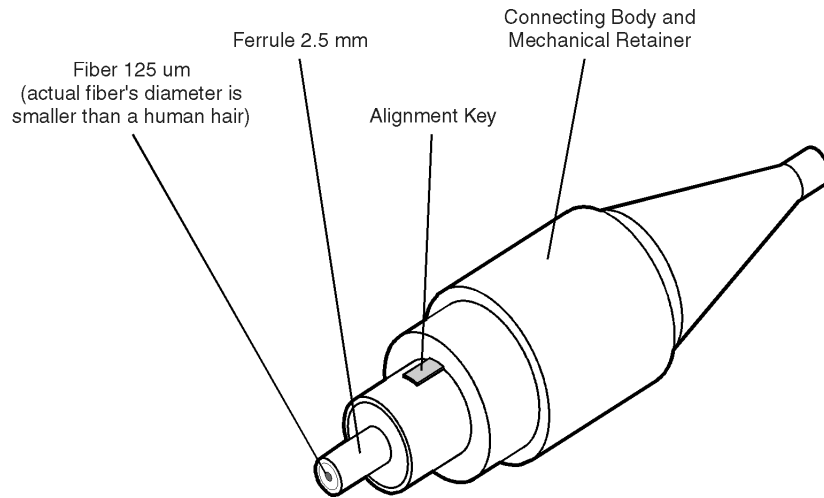
Choosing the Right Connector

A critical but often overlooked factor in making a good lightwave measurement is the selection of the fiber-optic connector. The differences in connector types are mainly in the mechanical assembly that holds the ferrule in position against another identical ferrule. Connectors also vary in the the polish, curve, and concentricity of the core within the cladding. Mating one style of cable to another requires an adapter. Agilent Technologies offers adapters for most instruments to allow testing with many different cables. The [Figure 1-1 on page 1-3](#) shows the basic components of a typical connectors.

The system tolerance for reflection and insertion loss must be known when selecting a connector from the wide variety of currently available connectors. Some items to consider when selecting a connector are:

- How much insertion loss can be allowed?
- Will the connector need to make multiple connections? Some connectors are better than others, and some are very poor for making repeated connections.
- What is the reflection tolerance? Can the system take reflection degradation?
- Is an instrument-grade connector with a precision core alignment required?
- Is repeatability tolerance for reflection and loss important? Do your specifications take repeatability uncertainty into account?
- Will a connector degrade the return loss too much, or will a fusion splice be required? For example, many DFB lasers cannot operate with reflections from connectors. Often as much as 90 dB isolation is needed.

Figure 1-1. Basic components of a connector.



Over the last few years, the FC/PC style connector has emerged as the most popular connector for fiber-optic applications. While not the highest performing connector, it represents a good compromise between performance, reliability, and cost. If properly maintained and cleaned, this connector can withstand many repeated connections.

However, many instrument specifications require tighter tolerances than most connectors, including the FC/PC style, can deliver. These instruments cannot tolerate connectors with the large non-concentricities of the fiber common with ceramic style ferrules. When tighter alignment is required, Agilent instruments typically use a connector such as the Diamond HMS-10, which has concentric tolerances within a few tenths of a micron. Agilent then uses a special universal adapter, which allows other cable types to mate with this precision connector. See [Figure 1-2](#).

Figure 1-2. Universal adapters to Diamond HMS-10



The HMS-10 encases the fiber within a soft nickel silver (Cu/Ni/Zn) center which is surrounded by a tough tungsten carbide casing, as shown in [Figure 1-3 on page 1-4](#).

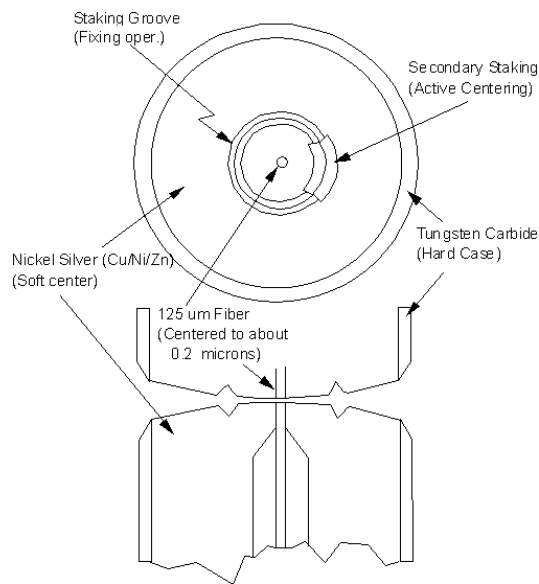


Figure 1-3. Cross-section of the Diamond HMS-10 connector.

The nickel silver allows an active centering process that permits the glass fiber to be moved to the desired position. This process first stakes the soft nickel silver to fix the fiber in a near-center location, then uses a post-active staking to shift the fiber into the desired position within 0.2 μm. This process, plus the keyed axis, allows very precise core-to-core alignments. This connector is found on most Agilent lightwave instruments.

The soft core, while allowing precise centering, is also the chief liability of the connector. The soft material is easily damaged. Care must be taken to minimize excessive scratching and wear. While minor wear is not a problem if the glass face is not affected, scratches or grit can cause the glass fiber to move out of alignment. Also, if unkeyed connectors are used, the nickel silver can be pushed onto the glass surface. Scratches, fiber movement, or glass contamination will cause loss of signal and increased reflections, resulting in poor return loss.

Inspecting Connectors

Because fiber-optic connectors are susceptible to damage that is not immediately obvious to the naked eye, bad measurements can be made without the user even being aware of a connector problem. Although microscopic examination and return loss measurements are the best way to ensure good connections, they are not always practical. An awareness of potential problems, along with good cleaning practices, can ensure that optimum connector performance is maintained. With glass-to-glass interfaces, it is clear that any degradation of a ferrule or the end of the fiber, any stray particles, or finger oil can have a significant effect on connector performance.

Figure 1-4 shows the end of a clean fiber-optic cable. The dark circle in the center of the micrograph is the fiber's 125 μm core and cladding which carries the light. The surrounding area is the soft nickel-silver ferrule. Figure 1-5 shows a dirty fiber end from neglect or perhaps improper cleaning. Material is smeared and ground into the end of the fiber causing light scattering and poor reflection. Not only is the precision polish lost, but this action can grind off the glass face and destroy the connector.

Figure 1-6 shows physical damage to the glass fiber end caused by either repeated connections made without removing loose particles or using improper cleaning tools. When severe, the damage on one connector end can be transferred to another good connector that comes in contact with it.

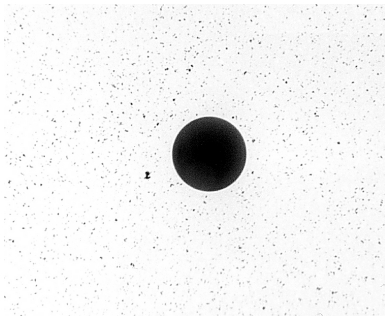


Figure 1-4. Clean, problem-free fiber end and ferrule.

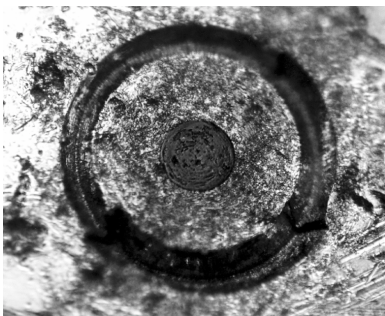


Figure 1-5. Dirty fiber end and ferrule from poor cleaning.



Figure 1-6. Damage from improper cleaning.

The cure for these problems is disciplined connector care as described in the following list and in [“Cleaning Connectors” on page 1-8.](#)

Use the following guidelines to achieve the best possible performance when making measurements on a fiber-optic system:

- Never use metal or sharp objects to clean a connector and never scrape the connector.
- Avoid matching gel and oils.

While these often work well on first insertion, they are great dirt magnets. The oil or gel grabs and holds grit that is then ground into the end of the fiber. Also, some early gels were designed for use with the FC, non-contacting connectors, using small glass spheres. When used with contacting connectors, these glass balls can scratch and pit the fiber. If an index matching gel or oil must be used, apply it to a freshly cleaned connector, make the measurement, and then immediately clean it off. Never use a gel for longer-term connections and never use it to improve a damaged connector. The gel can mask the extent of damage and continued use of a damaged fiber can transfer damage to the instrument.

- When inserting a fiber-optic cable into a connector, gently insert it in as straight a line as possible. Tipping and inserting at an angle can scrape material off the inside of the connector or even break the inside sleeve of connectors made with ceramic material.
- When inserting a fiber-optic connector into a connector, make sure that the fiber end does not touch the outside of the mating connector or adapter.
- Avoid over tightening connections.

Unlike common electrical connections, tighter is *not* better. The purpose of the connector is to bring two fiber ends together. Once they touch, tightening only causes a greater force to be applied to the delicate fibers. With connectors that have a convex fiber end, the end can be pushed off-axis resulting in misalignment and excessive return loss. Many measurements are actually improved by backing off the connector pressure. Also, if a piece of grit does happen to get by the cleaning procedure, the tighter connection is more likely to damage the glass. Tighten the connectors just until the two fibers touch.

- Keep connectors covered when not in use.
- Use fusion splices on the more permanent critical nodes. Choose the best connector possible. Replace connecting cables regularly. Frequently measure the return loss of the connector to check for degradation, and clean every connector, every time.

All connectors should be treated like the high-quality lens of a good camera. The weak link in instrument and system reliability is often the inappropriate use and care of the connector. Because current connectors are so easy to use, there tends to be reduced vigilance in connector care and cleaning. It takes only one missed cleaning for a piece of grit to permanently damage the glass and ruin the connector.

Measuring insertion loss and return loss

Consistent measurements with your lightwave equipment are a good indication that you have good connections. Since return loss and insertion loss are key factors in determining optical connector performance they can be used to determine connector degradation. A smooth, polished fiber end should produce a good return-loss measurement. The quality of the polish establishes the difference between the “PC” (physical contact) and the “Super PC” connectors. Most connectors today are physical contact which make glass-to-glass connections, therefore it is critical that the area around the glass core be clean and free of scratches. Although the major area of a connector, excluding the glass, may show scratches and wear, if the glass has maintained its polished smoothness, the connector can still provide a good low level return loss connection.

If you test your cables and accessories for insertion loss and return loss upon receipt, and retain the measured data for comparison, you will be able to tell in the future if any degradation has occurred. Typical values are less than 0.5 dB of loss, and sometimes as little as 0.1 dB of loss with high performance connectors. Return loss is a measure of reflection: the less reflection the better (the larger the return loss, the smaller the reflection). The best physically contacting connectors have return losses better than 50 dB, although 30 to 40 dB is more common.

To Test Insertion Loss

Use an appropriate lightwave source and a compatible lightwave receiver to test insertion loss. Examples of test equipment configurations include the following equipment:

- HP/Agilent 71450A or 71451A Optical Spectrum Analyzers with Option 002 built-in white light source.
- HP/Agilent 8702 or 8703 Lightwave Component Analyzer system.
- HP/Agilent 83420 Chromatic Dispersion Test Set with an HP/Agilent 8510 analyzer.
- HP/Agilent 8153 Lightwave Multimeter with a source and power sensor module.

To Test Return Loss

Use an appropriate lightwave source, lightwave receiver, and lightwave coupler to test return loss. Examples of test equipment configurations include the following equipment:

- HP/Agilent 8703 Lightwave Component Analyzer.
- HP/Agilent 8702 Lightwave Component Analyzer with the appropriate source, receiver, and lightwave coupler.
- HP/Agilent 8504 Precision Reflectometer.
- HP/Agilent 8153 Lightwave Multimeter with a source and power sensor module in conjunction with a lightwave coupler.
- HP/Agilent 81554SM Dual Source and HP/Agilent 81534A Return Loss Module.

Visual inspection of fiber ends

Visual inspection of fiber ends can be helpful. Contamination or imperfections on the cable end face can be detected as well as cracks or chips in the fiber itself. Use a microscope (100X to 200X magnification) to inspect the entire end face for contamination, raised metal, or dents in the metal as well as any other imperfections. Inspect the fiber for cracks and chips. Visible imperfections not touching the fiber core may not affect performance (unless the imperfections keep the fibers from contacting).

WARNING Always remove both ends of fiber-optic cables from any instrument, system, or device before visually inspecting the fiber ends. Disable all optical sources before disconnecting fiber-optic cables. Failure to do so may result in permanent injury to your eyes.

Cleaning Connectors

The procedures in this section provide the proper steps for cleaning fiber-optic cables and Agilent universal adapters. The initial cleaning, using the alcohol as a solvent, gently removes any grit and oil. If a caked-on layer of material is still present, (this can happen if the beryllium-copper sides of the ferrule retainer get scraped and deposited on the end of the fiber during insertion of the cable), a second cleaning should be performed. It is not uncommon for a cable or connector to require more than one cleaning.

CAUTION Agilent Technologies strongly recommends that index matching compounds *not* be applied to their instruments and accessories. Some compounds, such as gels, may be difficult to remove and can contain damaging particulates. If you think the use of such compounds is necessary, refer to the compound manufacturer for information on application and cleaning procedures.

Table 1-1. Cleaning Accessories

Item	Agilent Part Number
Pure isopropyl alcohol	—
Cotton swabs	8520-0023
Small foam swabs	9300-1223
Compressed dust remover (non-residue)	8500-5262

Table 1-2. Dust Caps Provided with Lightwave Instruments

Item	Agilent Part Number
Laser shutter cap	08145-64521
FC/PC dust cap	08154-44102
Biconic dust cap	08154-44105
DIN dust cap	5040-9364
HMS10/Agilent dust cap	5040-9361
ST dust cap	5040-9366

To clean a non-lensed connector

CAUTION Do not use any type of foam swab to clean optical fiber ends. Foam swabs can leave filmy deposits on fiber ends that can degrade performance.

1. Apply pure isopropyl alcohol to a clean lint-free cotton swab or lens paper.
Cotton swabs can be used as long as no cotton fibers remain on the fiber end after cleaning.
2. Clean the ferrules and other parts of the connector while avoiding the end of the fiber.
3. Apply isopropyl alcohol to a new clean lint-free cotton swab or lens paper.
4. Clean the fiber end with the swab or lens paper.
5. Do *not* scrub during this initial cleaning because grit can be caught in the swab and become a gouging element.
6. Immediately dry the fiber end with a clean, dry, lint-free cotton swab or lens paper.
7. Blow across the connector end face from a distance of 6 to 8 inches using filtered, dry, compressed air. Aim the compressed air at a shallow angle to the fiber end face.
8. Nitrogen gas or compressed dust remover can also be used.

CAUTION Do not shake, tip, or invert compressed air canisters, because this releases particles in the can into the air. Refer to instructions provided on the compressed air canister.

9. As soon as the connector is dry, connect or cover it for later use.

If the performance, after the initial cleaning, seems poor try cleaning the connector again. Often a second cleaning will restore proper performance. The second cleaning should be more arduous with a scrubbing action.

To clean an adapter

The fiber-optic input and output connectors on many Agilent instruments employ a universal adapter such as those shown in the following picture. These adapters allow you to connect the instrument to different types of fiber-optic cables.

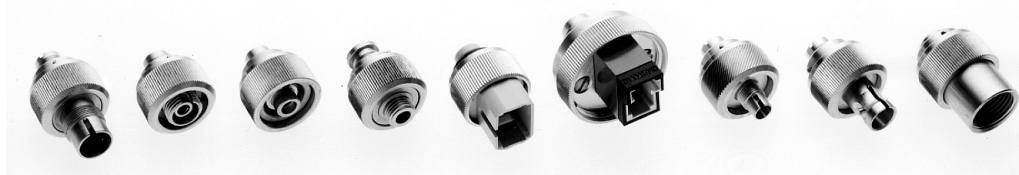


Figure 1-7. Universal adapters

1. Apply isopropyl alcohol to a clean foam swab.

Cotton swabs can be used as long as no cotton fibers remain after cleaning. The foam swabs listed in this section's introduction are small enough to fit into adapters.

Although foam swabs can leave filmy deposits, these deposits are very thin, and the risk of other contamination buildup on the inside of adapters greatly outweighs the risk of contamination by foam swabs.

2. Clean the adapter with the foam swab.
3. Dry the inside of the adapter with a clean, dry, foam swab.
4. Blow through the adapter using filtered, dry, compressed air.

Nitrogen gas or compressed dust remover can also be used. Do not shake, tip, or invert compressed air canisters, because this releases particles in the can into the air. Refer to instructions provided on the compressed air canister.

Caring for Electrical Connections

The following list includes the basic principles of microwave connector care. For more information on microwave connectors and connector care, consult the *Hewlett-Packard Microwave Connector Care Manual*, HP part number 08510-90064.

Handling and Storage

- Keep connectors clean
- Extend sleeve or connector nut
- Use plastic endcaps during storage
- Do *not* touch mating plane surfaces
- Do *not* set connectors contact-end down

Visual Inspection

- Inspect all connectors carefully before every connection
- Look for metal particles, scratches, and dents
- Do *not* use damaged connectors

Cleaning

- Try cleaning with compressed air first
- Clean the connector threads
- Do *not* use abrasives
- Do *not* get liquid onto the plastic support beads

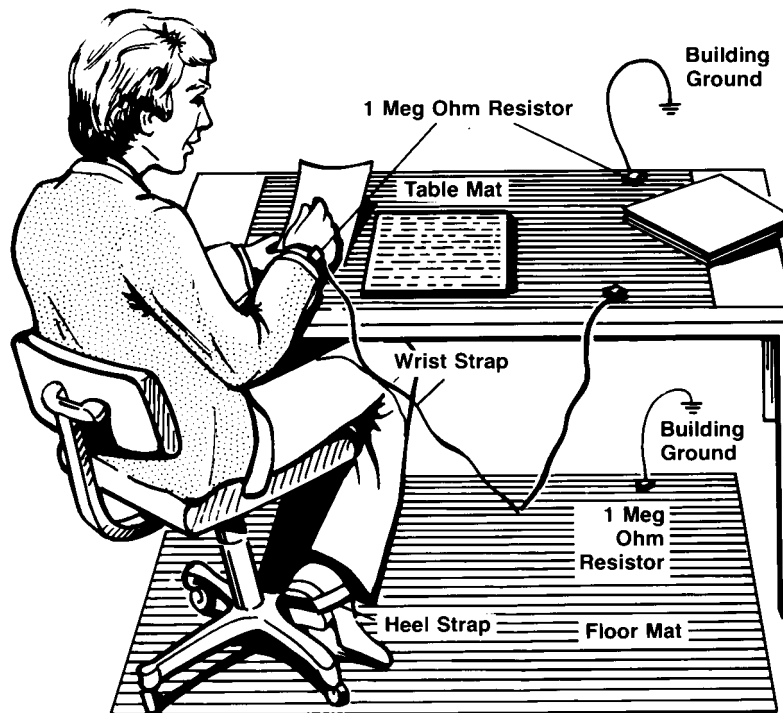
Making Connections

- Align connectors carefully
- Make preliminary connection lightly
- To tighten, turn connector nut *only*
- Do *not* apply bending force to connection
- Do *not* overtighten preliminary connection
- Do *not* twist or screw in connectors
- Do *not* tighten past the “break” point of the torque wrench

Electrostatic Discharge Information

Electrostatic discharge (ESD) can damage or destroy electronic components. All work on electronic assemblies should be performed at a static-safe work station. The following figure shows an example of a static-safe work station using two types of ESD protection:

- Conductive table-mat and wrist-strap combination.
- Conductive floor-mat and heel-strap combination.



Both types, when used together, provide a significant level of ESD protection. Of the two, only the table-mat and wrist-strap combination provides adequate ESD protection when used alone.

To ensure user safety, the static-safe accessories must provide at least 1 M Ω of isolation from ground. Refer to [Table 1-3 on page 1-13](#) for information on ordering static-safe accessories.

WARNING These techniques for a static-safe work station should not be used when working on circuitry with a voltage potential greater than 500 volts.

Reducing ESD Damage

The following suggestions may help reduce ESD damage that occurs during testing and servicing operations.

- Personnel should be grounded with a resistor-isolated wrist strap before removing any assembly from the unit.
- Be sure all instruments are properly earth-grounded to prevent a buildup of static charge.

Table 1-3. Static-Safe Accessories

Agilent Part Number	Description
9300-0797	Set includes: 3M static control mat 0.6 m × 1.2 m (2 ft× 4 ft) and 4.6 cm (15 ft) ground wire. (The wrist-strap and wrist-strap cord are not included. They must be ordered separately.)
9300-0980	Wrist-strap cord 1.5 m (5 ft)
9300-1383	Wrist-strap, color black, stainless steel, without cord, has four adjustable links and a 7 mm post-type connection.
9300-1169	ESD heel-strap (reusable 6 to 12 months).

2

- “Making Measurements” on page 2-2
- “Making a Basic Measurement” on page 2-3
- “Using Display Functions” on page 2-4
- “Using Markers” on page 2-17
- “Using the List Mode to Test a Device” on page 2-37
- “Using Limit Lines to Test a Device” on page 2-42
- “Using Ripple Limits to Test a Device” on page 2-50
- “Using Bandwidth Limits to Test a Bandpass Filter” on page 2-60
- “Using Test Sequencing” on page 2-66
- “Using Test Sequencing to Test a Device” on page 2-77
- “Magnitude and Phase Comparisons of Lightwave Receivers” on page 2-83
- “Mathematically Combining Device Responses” on page 2-91
- “Using a Coefficient Model for Device Response and Design Model Comparison” on page 2-95
- “Using an External Laser” on page 2-98

Making Measurements

Making Measurements

This chapter contains an outline of the process for measuring a device and example procedures for making various measurements. This chapter also describes how to use display, marker, and sequencing functions.

CAUTION Use of controls, or adjustment, or performance of procedures other than those specified herein may result in hazardous radiation exposure.

Making a Basic Measurement

There are four basic steps when you are making a measurement.

Table 2-1. Basic Measurement Sequence

<p>Step 1. Set Up Measurement</p>	<ul style="list-style-type: none"> • Connect device under test, including all adapters and cables. • Select measurement type: E/O, O/O, O/E. • Select settings: start and stop frequencies, power, number of points, IF bandwidth, averaging, sweep type, trigger.
<p>Step 2. Perform a Calibration</p>	<ul style="list-style-type: none"> • Select calibration type: response, response and isolation, response and match. • Measure calibration standards. • Save calibration data.
<p>Step 3. Measure the Device Response</p>	<ul style="list-style-type: none"> • Reconnect device under test. • Analyze measurement data: markers, limit testing. • Perform any math operations.
<p>Step 4. Output Measurement Results</p>	<ul style="list-style-type: none"> • Save measurement setup and data. • Print measurement results.

←

Using Display Functions

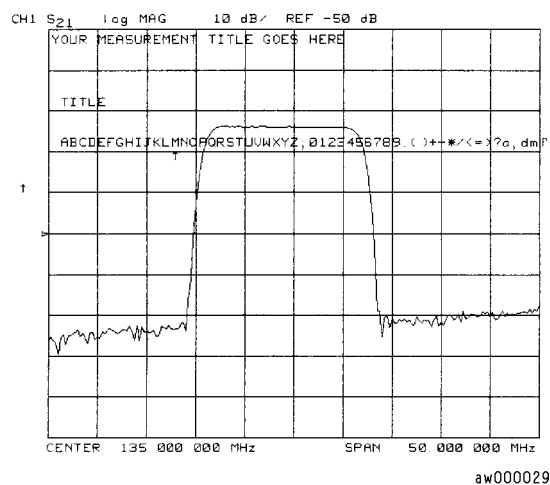
This section provides information for using the display functions. These functions are helpful for displaying measurement data so that it will be easy to read. This section covers the following topics:

- Adding titles to your measurements
- Viewing both primary channels at the same time
- Viewing and customizing four-channel measurements
- Using the memory traces
- Using the memory math functions
- Blanking the analyzer's display
- Changing the colors of the display

Titling the Active Channel Display

1. Press **Display**, **MORE**, **TITLE** to access the title menu.
2. Press **ERASE TITLE** and enter the title you want for your measurement display.
 - If you have a DIN keyboard attached to the analyzer, type the title you want from the keyboard. Then press **ENTER** to enter the title into the analyzer. You can enter a title that has a maximum of 50 characters. (For more information on using a keyboard with the analyzer, refer to the “Options and Accessories” chapter in the reference guide.)
 - If you do not have a DIN keyboard attached to the analyzer, enter the title from the analyzer front panel.
 - a. Turn the front panel knob to move the arrow pointer to the first character of the title.
 - b. Press **SELECT LETTER**.
 - c. Repeat the previous two steps to enter the rest of the characters in your title. You can enter a title that has a maximum of 50 characters.
 - d. Press **DONE**, to complete the title entry.

Figure 2-1. Example of a Display Title



CAUTION The **NEWLINE**, and **FORMFEED**, keys are not intended for creating display titles. Those keys are for creating commands to send to peripherals during a sequence program.

Viewing Both Primary Measurement Channels

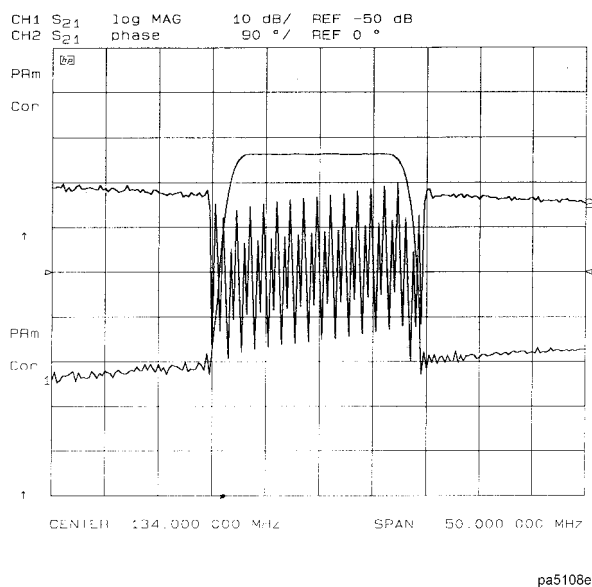
In some cases, you may want to view more than one measured parameter at a time. Simultaneous gain and phase measurements, for example, are useful in evaluating stability in negative feedback amplifiers. You can easily make such measurements using the dual channel display.

1. To see channels 1 and 2 in the same grid, press:

Display, DUAL | QUAD SETUP

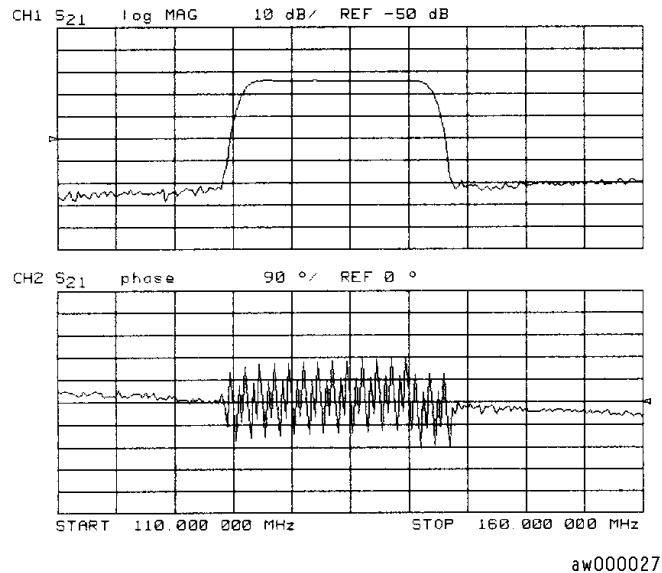
set **DUAL CHAN** on **OFF** to **ON**, and **SPLIT DISP** to **1X**

Figure 2-2. Example of Viewing Channel 1 and 2 Simultaneously



2. To view the measurements on separate graticules, press: Set **SPLIT DISP** to **2X**. The analyzer shows channel 1 on the upper half of the display and channel 2 on the lower half of the display. The analyzer defaults to measuring O/O on channel 1 and channel 2.

Figure 2-3. Example Dual Channel with Split Display On



3. To return to a single-graticule display, press: **SPLIT DISPLAY 1X**.

Dual Channel Mode with Decoupled Stimulus

The stimulus functions of the two channels can be controlled independently using **COUPLED CH ON off** in the sweep setup menu. In addition, the markers can be controlled independently for each channel using **MARKERS: UNCOUPLED** in the marker mode menu, under the **Marker Fctn** key.

NOTE For dual channel, if channels are uncoupled and you have full 2-port calibrations on both channels, you will not be able to select a non-ratioed measurement. For example, you can measure S_{21} or B/R, but not input B.

NOTE Auxiliary channels 3 and 4 are permanently coupled by stimulus to primary channels 1 and 2, respectively. Decoupling the primary channels' stimulus from each other does not affect the stimulus coupling between the auxiliary channels and their primary channels.

Dual Channel Mode with Decoupled Channel Power

By decoupling the channel power or port power and using the dual channel mode, you can simultaneously view two measurements (or two sets of measurements, if both auxiliary channels are enabled) having different power levels.

However, there are two configurations that will not sweep continuously.

1. For analyzers with source attenuators, with channel 1 having one attenuation value and channel 2 set to a different attenuation value, then continuous sweep is disabled to avoid

wear on the attenuator. A similar situation where this occurs is when a 2-port cal is active and the port 1 attenuation value is not equal to the port 2 attenuation value. Since one attenuator is used for both measurements, this would cause the attenuator to continuously switch power ranges, so continuous sweep is not allowed.

2. Channel 1 is driving one test port and channel 2 is driving the other test port. This would cause the test port transfer switch to continually cycle. The instrument will not allow the transfer switch or attenuator to continuously switch ranges in order to update these measurements without the direct intervention of the operator.

If one of these conditions exist, the test set hold mode will engage, and the status notation τ_{SH} will appear on the left side of the screen. The hold mode leaves the measurement function in only one of the two measurements. To update both measurement setups, press **Sweep Setup, MEASURE RESTART**.

Viewing Four Measurement Channels

Four measurement channels can be viewed simultaneously by enabling auxiliary channels 3 and 4. Although independent of other channels in most variables, channels 3 and 4 are permanently coupled to channels 1 and 2 respectively by stimulus. That is, if channel 1 is set for a center frequency of 200 MHz and a span of 50 MHz, channel 3 will have the same stimulus values.

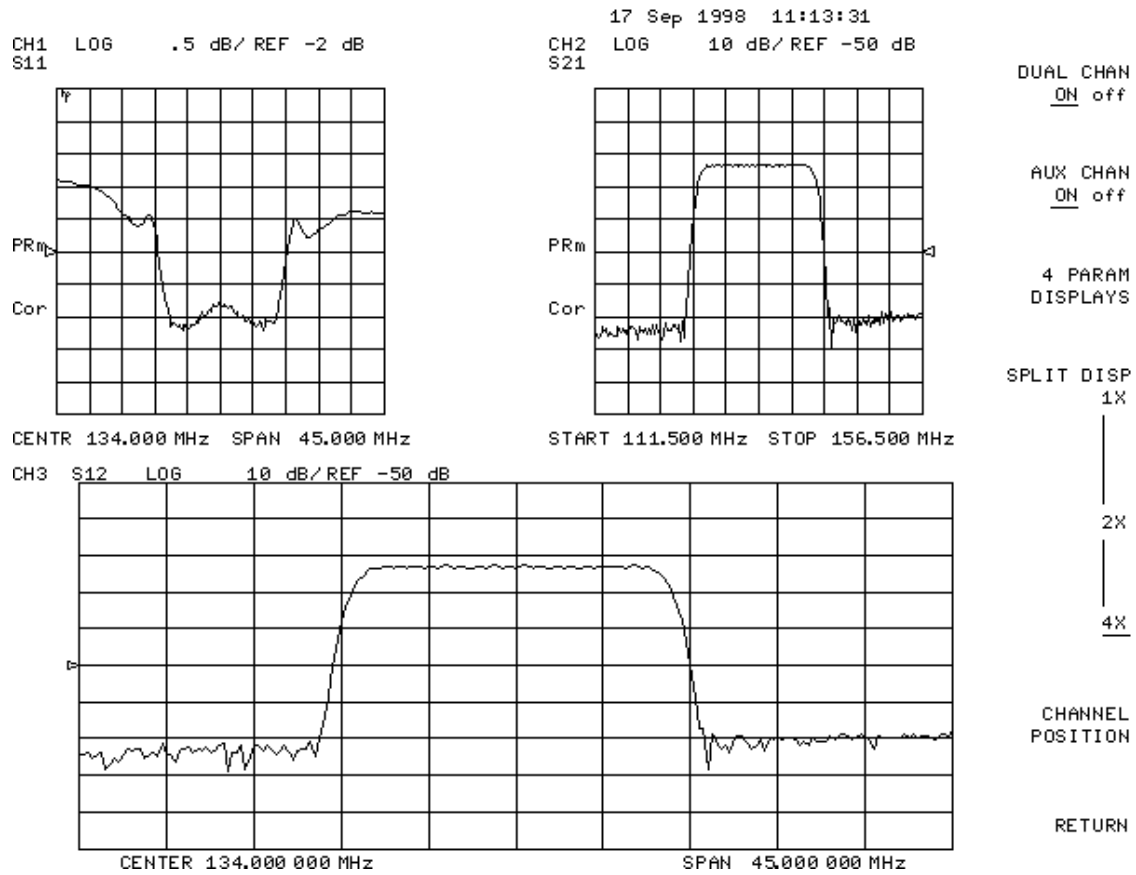
NOTE Channels 1 and 2 are referred to as primary channels and channels 3 and 4 are referred to as auxiliary channels.

Channel 3 or 4 are activated when the Chan 3 or Chan 4 keys are pressed. Alternatively, you can enable the auxiliary channels by setting **AUX CHAN** to **ON**. For example, if channel 1 is active, pressing **AUX CHAN**, to ON enables channel 3 and its trace appears on the display. Channel 4 is similarly enabled and viewed when channel 2 is active.

1. Press **Format** to select the type of display of the data. This example uses the log mag format.
2. If channel 1 is not active, make it active by pressing **Chan 1**.
3. Press **Display, DUAL | QUAD SETUP**, set **DUAL CHAN**, to ON, set **AUX CHAN**, to ON, and set **SPLIT DISP**, to **4X**.

The display will appear as shown in [Figure 2-4 on page 2-9](#). Channel 1 is in the upper-left quadrant of the display, channel 2 is in the upper-right quadrant, and channel 3 is in the lower half of the display.

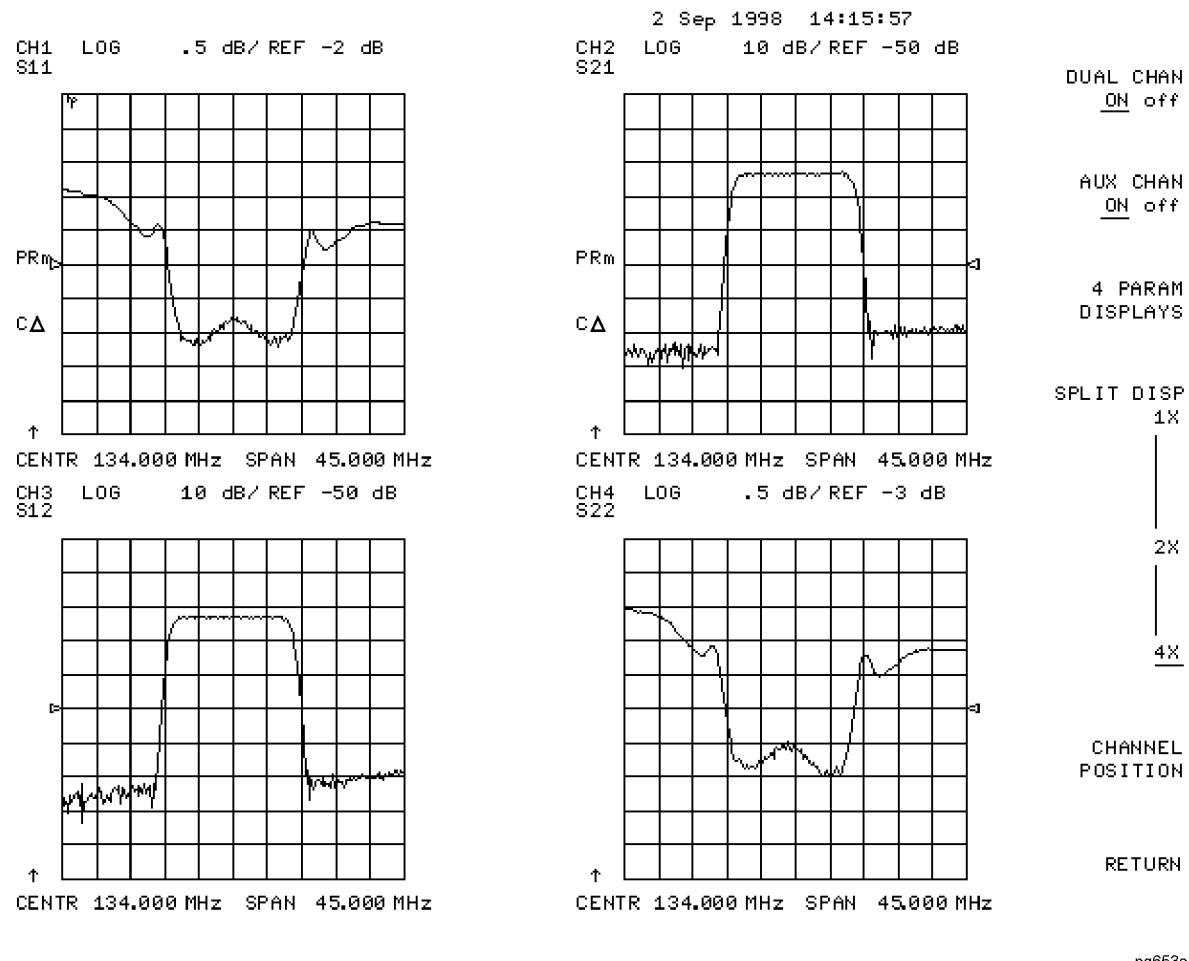
Figure 2-4. Three-Channel Display



4. Press Chan 4 (or press **Chan 2**, set **AUX CHAN**, to ON).

This enables channel 4 and the screen now displays four separate grids as shown in [Figure 2-5 on page 2-10](#). Channel 4 is in the lower-right quadrant of the screen.

Figure 2-5. Four-Channel Display



5. Press **Chan 4**.

Observe that the amber LED adjacent to the **Chan 4** key is lit and the CH4 indicator on the display has a box around it. This indicates that channel 4 is now active and can be configured.

6. Press **Marker, MARKER 1, MARKER 2**.

Markers 1 and 2 appear on all four channel traces. Rotating the front panel control knob moves marker 2 on all four channel traces. Note that the active function, in this case the marker frequency, is the same color and in the same grid as the active channel (channel 4).

7. Press **Chan 3**.

Observe that the amber LED adjacent to the **Chan 3** key is lit. This indicates that channel 3 is now active and can be configured.

8. Rotate the front panel control knob and notice that marker 2 still moves on all four channel traces.

9. To independently control the channel markers:

Press **Marker Fctn**, **MORE**, **MARKER MODE MENU**, set **MARKERS**: to **UNCOUPLED**.

Rotate the front panel control knob. Marker 2 moves only on the channel 3 trace.

Once made active, a channel can be configured independently of the other channels in most variables except stimulus. For example, once channel 3 is active, you can change its format to a Smith chart by pressing **Format**, **SMITH CHART**.

Customizing the Four-Channel Display

When one or both auxiliary channels are enabled, **DUAL CHAN** on **OFF**, and **SPLIT DISP 1X 2X 4X**, interact to produce different display configurations according to [Table 2-2](#).

Table 2-2. Customizing the Display

Split Display	Dual Channel	Aux Channels On	Number of Graticules
1X	Don't Care	Don't Care	1
1X/2X/4X	Off	None	
2X/4X	Off	3 or 4	2
2X	On	Don't Care	
4X	On	3 or 4	3
4X	On	Both on	4

Channel Position Softkey

CHANNEL POSITION, gives you options for arranging the display of the channels. Press **Display**, **DUAL|QUAD SETUP**, to use **CHANNEL POSITION**.

CHANNEL POSITION, works with **SPLIT DISP 1X 2X 4X**. When **SPLIT DISP 2X**, is selected, **CHANNEL POSITION**, gives you two choices for a two-graticule display:

- Channels 1 and 2 overlaid in the top graticule, and channels 3 and 4 are overlaid in the bottom graticule.
- Channels 1 and 3 are overlaid in the top graticule, and channels 2 and 4 are overlaid in the bottom graticule.

When **SPLIT DISP 4X**, is selected, **CHANNEL POSITION**, gives you two choices for a four-graticule display:

- Channels 1 and 2 are in separate graticules in the upper half of the display, channels 3 and 4 are in separate graticules in the lower half of the display.
- Channels 1 and 3 are in the upper half of the display, channels 2 and 4 are in the lower half of the display.

4 Param Displays Softkey

The **4 PARAM DISPLAYS**, menu does two things:

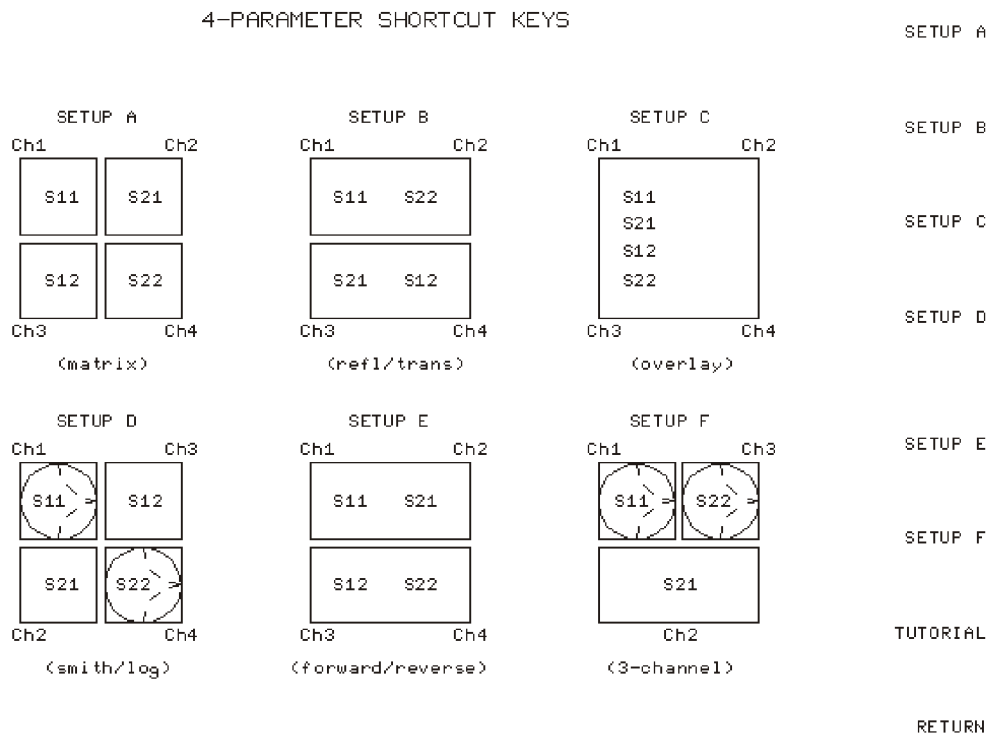
- provides a quick way to set up a four-parameter display
- gives information for using softkeys in the **Display**, menu

Figure 2-6 shows the first **4 PARAM DISPLAYS**, screen. Six setup options are described with softkeys **SETUP A**, through **SETUP F**. **SETUP A** is a four-parameter display where each channel is displayed on its own grid. Pressing **SETUP A** immediately produces a four-grid, four-parameter display. **SETUP B**, is also a four-parameter display, except that channel 1 and channel 2 are overlaid on the upper grid and channel 3 and channel 4 are overlaid on the lower grid. The other setup softkeys operate similarly. Notice that setups D and F produce displays which include Smith charts.

Pressing **TUTORIAL**, opens a screen which lists the order of keystrokes you would have to enter in order to create some of the setups without using one of the setup softkeys. The keystroke entries are listed (from top to bottom) beneath each setup and are color-coded to show the relationship between the keys and the channels. For example, beneath the four-grid display, [CHAN 1] and [MEAS] S11 are shown in yellow. Notice that in the four-grid graphic, Ch1 is also yellow, indicating that the keys in yellow apply to channel 1.

Pressing **MORE HELP**, opens a screen which lists the hardkeys and softkeys associated with the auxiliary channels and setting up multiple-channel, multiple-grid displays. Next to each key is a description of its function.

Figure 2-6. 4 Param Displays Menu



Using Memory Traces and Memory Math Functions

The analyzer has four available memory traces, one per channel. Memory trace 1 and memory trace 2 can be used with either channel 1 or channel 2. Memory trace 3 is only associated with channel 3, and memory trace 4 is only associated with channel 4.

Memory traces can be saved with instrument states: one memory trace can be saved per channel for each saved instrument state. There are up to 31 save/recall registers available, so the total number of memory traces that can be present is 128 including the four active for the current instrument state. The memory data is stored as full precision, complex data. Memory traces must be displayed in order to be saved with instrument states.

Additional data can be stored onto 3.5-inch floppy disks using the front panel disk drive.

NOTE You may not be able to store 31 instrument states if they include a large amount of calibration data. The calibration data contributes considerably to the size of the instrument state file and therefore the available memory may be full prior to filling all 31 registers. [“Memory Allocation” on page 8-13](#) in the Reference.

The following trace math operations are implemented:

Table 2-3.

DATA/MEM	MEM1/MEM2
DATA-MEM	MEM1-MEM2
DATA*MEM	MEM1*MEM2
DATA+MEM	MEM2/MEM1
MEM/DATA	MEM2-MEM1
MEM-DATA	

Memory traces are saved and recalled and trace math is performed immediately after error-correction. This means that any data processing done after error-correction, including parameter conversion, scaling, and so forth, can be performed on the memory trace. You can also use trace math as a simple means of error-correction, although that is not its main purpose.

All data processing operations that occur after trace math, except smoothing is identical for the data trace and the memory trace. If smoothing is on when a memory trace is saved, this state is maintained regardless of the data trace smoothing. If a memory trace is saved with smoothing on, this feature can be turned on or off in the memory-only display mode.

The actual memory for storing a memory trace is allocated only as needed. The memory trace is cleared on instrument preset, power on, or instrument state recall.

If sweep mode or sweep range is different between the data and memory traces, trace math is allowed, and no warning message is displayed. If the number of points in the two traces is

different, the memory trace is not displayed nor rescaled. However, if the number of points for the data trace is changed back to the number of points in the memory, the memory trace can then be displayed.

If trace math or display memory is requested and no memory trace exists, the message CAUTION: NO VALID MEMORY TRACE is displayed.

To Save a Data Trace to the Display Memory

Press **Display, DATA→MEMORY**, to store the current active measurement data in the memory of the active channel. The data trace is now also the memory trace. You can use a memory trace for subsequent math manipulations.

To View the Measurement Data and Memory Trace

The analyzer default setting shows you the current measurement data for the active channel.

1. To view a data trace that you have already stored to the active channel memory, press:

Display, MEMORY

This is the only memory display mode where you can change the smoothing of the memory trace.

2. To view both the memory trace and the current measurement data trace, press:

Display, DATA and MEMORY

To Perform a Math Function

1. You must have already stored a data trace to the active channel memory, as described in [“To Save a Data Trace to the Display Memory” on page 2-14](#).
2. Press **Display, MATH:DATA/MEM** to divide the data by the memory.
3. To perform other math functions on the measurement data and memory, use the menu under the **MATH:DATA/MEM** key.

Blanking the Display

Pressing **Display, MORE, ADJUST DISPLAY, BLANK DISPLAY** switches off the analyzer display while leaving the instrument in its current measurement state. This feature may be helpful in prolonging the life of the LCD in applications where the analyzer is left unattended (such as in an automated test system). Turning the front panel knob or pressing any front panel key will restore normal display operation.

Pressing **Display, FREQUENCY BLANK** will blank the displayed frequency notation for security purposes. The frequency labels cannot be restored except by instrument preset or turning the power off and then on.

Adjusting the Colors of the Display

Setting Display Intensity

To adjust the intensity of the display, press **Display, MORE, ADJUST DISPLAY, INTENSITY**, and rotate the front panel knob, use the (**up**,) (**down**,) keys, or use the numerical keypad to set the intensity value between 50 and 100 percent. Lowering the intensity may prolong the life of the LCD.

Setting Default Colors

To set all the display elements to the factory-defined default colors, press **DISPLAY, MORE ADJUST DISPLAY, DEFAULT COLORS**.

NOTE **Preset**, does not reset or change colors to the default color values. However, cycling power to the instrument will reset the colors to the default color values.

The Modify Colors Menu

The **MODIFY COLORS** softkey within the adjust display menu provides access to the modify colors menu.

The modify colors menu allows you to adjust the colors on your analyzer's display. The default colors in this instrument were chosen to maximize your ability to discern the difference between the channel colors, and to comfortably and effectively view the colors. Each channel's memory trace color was chosen because the color is similar to the channels data trace color. This allows easy association between the data trace and the memory trace for each channel.

You may choose to change the default colors to suit environmental needs, individual preferences, or to accommodate color deficient vision. You can use any of the available colors for any of the display elements listed:

CH1 DATA/LIMIT LN	CH3 DATA/LIMIT LN
CH1 MEM	CH3 MEM
CH2 DATA/LIMIT LN	CH4 DATA/LIMIT LN
CH2 MEM	CH4 MEM
GRATICULE	REF LINE
TEXT	WARNING
RIPPLE LIM LINES	

To change the color of a display element, press the softkey for that element (such as **CH1 DATA**). Then press **TINT** and turn the analyzer front panel knob; use the step keys or the numeric keypad, until the desired color appears.

NOTE Maximum viewing with the LCD display is achieved when primary colors or a combination of them are selected at full brightness (100%). [Table 2-4](#) lists the recommended colors and their corresponding tint numbers.

Table 2-4. Display Colors with Maximum Viewing Angle

Display Color	Tint	Brightness	Color
Red	0	100	100
Yellow	17	100	100
Green	33	100	100
Cyan	50	100	100
Blue	67	100	100
Magenta	83	100	100
White	N/A	100	0

Color is comprised of three parameters:

- **Tint:** The continuum of hues on the color wheel, ranging from red, through green and blue, and back to red.
- **Brightness:** A measure of the brightness of the color.
- **Color:** The degree of whiteness of the color. A scale from white to pure color.

The most frequently occurring color deficiency is the inability to distinguish red, yellow, and green from one another. Confusion between these colors can usually be eliminated by increasing the brightness between the colors. To accomplish this, press the **BRIGHTNESS** softkey and turn the analyzer front panel knob. If additional adjustment is needed, vary the degree of whiteness of the color. To accomplish this, press the **COLOR**, softkey and turn the analyzer front panel knob.

NOTE Color changes and adjustments remain in effect until changed again in these menus or the analyzer is powered off and then on again. Cycling the power changes all color adjustments to default values. Once the colors are saved, pressing the **Preset** key does not affect the color selections.

Saving Modified Colors

To save a modified color set, press **Display, MORE, ADJUST DISPLAY, SAVE COLORS**. Modified colors are not part of a saved instrument state and are lost unless saved using these softkeys. Once modified colors are saved, they will be the colors applied until you press **Preset**.

Recalling Modified Colors

To recall the previously saved color set, press **RECALL COLORS**.

Using Markers

The **Marker**, key displays a movable active marker on the screen and provides access to a series of menus to control up to five display markers for each channel. By using markers, you can obtain numerical readings of measured values. Markers also provide capabilities for reducing measurement time by changing stimulus parameters, searching the trace for specific values, or statistically analyzing part or all of the trace.

Markers have a stimulus value (the x-axis value in a Cartesian format) and a response value (the y-axis value in a Cartesian format). In polar format, the second part of a complex data pair is also provided as an auxiliary response value. In Smith chart format, the real and imaginary rectangle are both displayed, and the effective capacitance or inductance of the imaginary part is also displayed. When a marker is activated and no other function is active, its stimulus value is displayed in the active entry area and can be controlled with the knob, the step keys, or the numeric keypad. The active marker can be moved to any point on the trace, and its response and stimulus values are displayed at the top right corner of the graticule for each displayed channel, in units appropriate to the display format. The displayed marker response values are valid even when the measured data is above or below the range displayed on the graticule.

- If you activate both data and memory traces, the marker values apply to the data trace.
- If you activate only the memory trace, the marker values apply to the memory trace.
- If you activate a memory math function, the marker values apply to the trace resulting from the memory math function.

Marker values are normally continuous: that is, they are interpolated between measured points. They can also be set to read only discrete measured points. Markers normally have the same stimulus values for all channels, or they can be uncoupled so that each channel has independent markers, regardless of whether stimulus values are coupled or dual channel display is on.

To Use Continuous and Discrete Markers

The analyzer can either place markers on discrete measured points, or move the markers continuously along a trace by interpolating the data value between measured points.

- Press **Marker Fctn**, **MORE**, **MARKER MODE MENU**, and select one of the following choices:
 - Choose **MARKERS: CONTINUOUS**, if you want the analyzer to place markers at any point on the trace, by interpolating between measured points. This default mode allows you to conveniently obtain round numbers for the stimulus value.
 - Choose **MARKERS: DISCRETE**, if you want the analyzer to place markers only on measured trace points determined by the stimulus settings. This may be the best mode to use with automated testing, using a computer or test sequencing because the analyzer does not interpolate between measured points.

NOTE Using **MARKERS: DISCRETE**, will also affect marker search and positioning functions when the value entered in a search or positioning function does not exist as a measurement point. The marker will be positioned to the closest adjacent point that satisfies the search or positioning value.

To Activate Display Markers

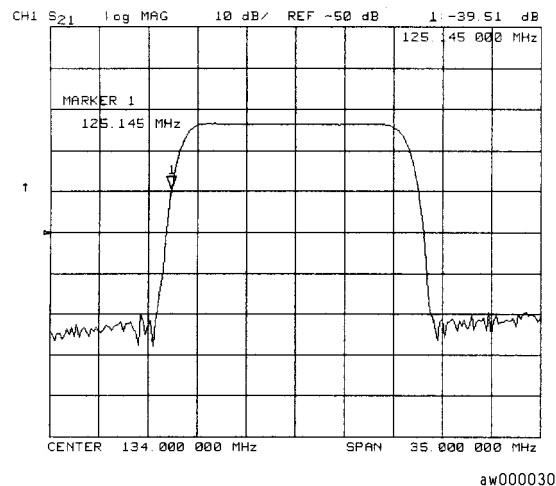
- To switch on marker 1 and make it the active marker, press:

Marker, MARKER 1

The active marker is identified on the analyzer display with the following symbol: ▽

The active marker stimulus value is displayed in the active entry area. You can modify the stimulus value of the active marker, using the front panel knob or numerical keypad. All of the marker response and stimulus values are displayed in the upper right corner of the display.

Figure 2-7. Active Marker Control Example

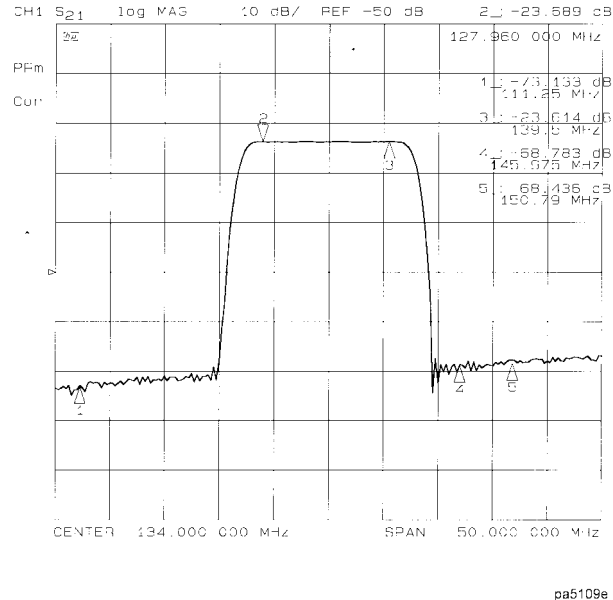


To switch on the corresponding marker and make it the active marker, press:

MARKER 2 or MARKER 3 or MARKER 4, or MARKER 5

All of the markers, other than the active marker, become inactive and are represented on the analyzer display as Δ . The active and inactive markers are shown in [Figure 2-8](#).

Figure 2-8. Active and Inactive Markers Example



- To switch off all of the markers, press **ALL OFF**.

To Move Marker Information Off the Grids

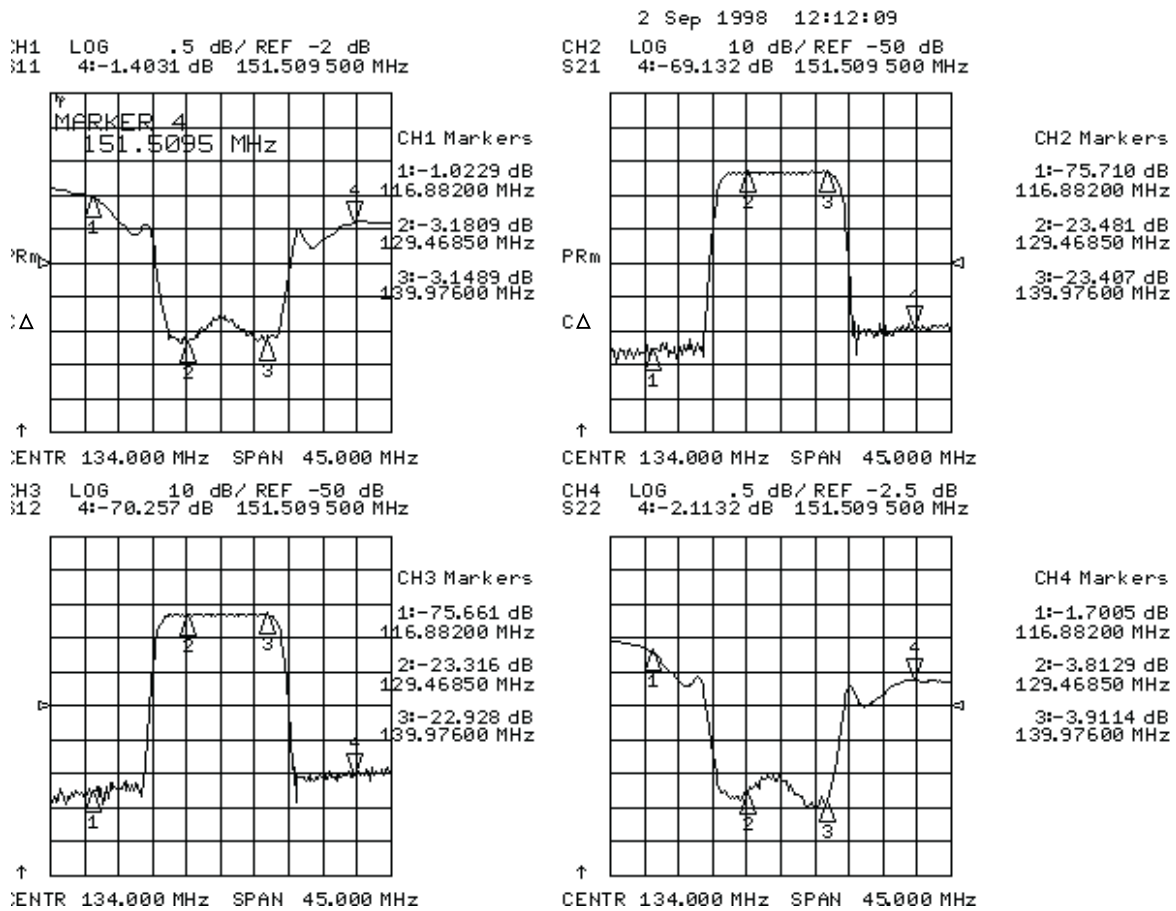
If marker information obscures the display traces, you can turn off the softkey menu and move the marker information off the display traces and into the softkey menu area. Pressing the backspace key \leftarrow , performs this function. This is a toggle function. Pressing \leftarrow , alternately hides and restores the current softkey menu. The softkey menu is also restored when you press any softkey or a hardkey which leads to a menu. The following procedure shows how you can view the marker information in the softkey area.

1. Set up a four-graticule display as described in "[Viewing Four Measurement Channels](#)" on [page 1-18](#).
2. Activate four markers by pressing **Marker, 1, 2, 3, 4**.

NOTE Observe that the markers appear on all of the grids. To activate markers on individual grids, press **Marker Fctn, MORE, MARKER MODE MENU**, and set **MARKERS:**, to **UNCOUPLED**. Then, activate the channel in which you wish to have markers, press **Marker** then select the markers for that channel.

3. Turn off the softkey menu and move the marker information off the grids by pressing \leftarrow . The display will be similar to [Figure 2-9](#).

Figure 2-9. Marker Information Moved into the Softkey Menu Area

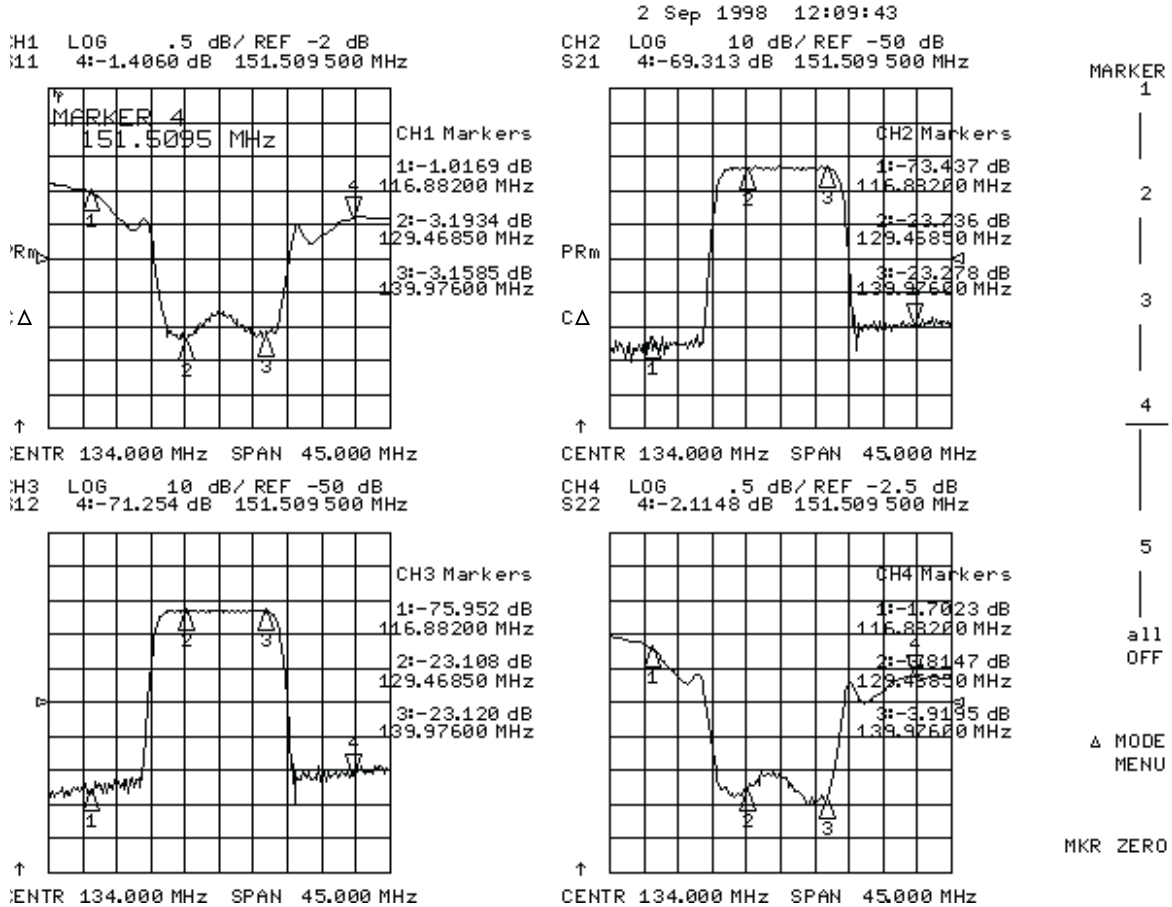


pg654e

- Restore the softkey menu and move the marker information back onto the graticules:
Press \leftarrow .

The display will be similar to [Figure 2-10](#).

Figure 2-10. Marker Information on the Graticules



pg655e

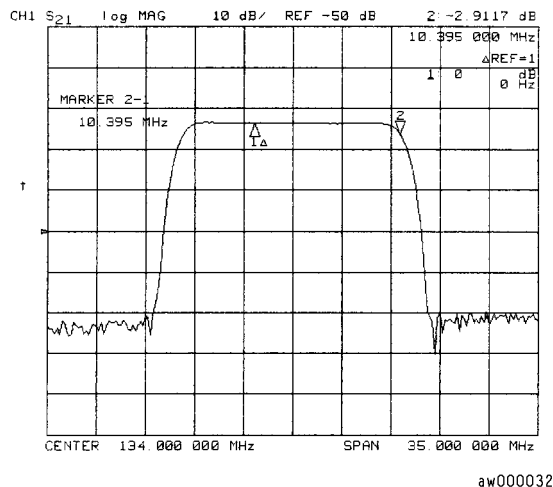
You can also restore the softkey menu by pressing a hardkey which opens a menu (such as **Meas**) or pressing a softkey.

To Use Delta (Δ) Markers

This is a relative mode, where the marker values show the position of the active marker relative to the delta reference marker. You can switch on the delta mode by defining one of the five markers as the delta reference.

1. Press **Marker**, **Δ MODE MENU**, **Δ REF=1**, to make marker 1 a reference marker.
2. To move marker 1 to any point that you want to reference:
 - Turn the front panel knob.
 - OR
 - Enter the frequency value (relative to the reference marker) on the numeric keypad.
3. Press **Marker**, **MARKER 2**, and move marker 2 to any position that you want to measure in reference to marker 1.

Figure 2-11. Marker 1 as the Reference Marker Example



4. To change the reference marker to marker 2, press:

Δ MODE MENU, Δ REF=2

To Activate a Fixed Marker

When a reference marker is fixed, it does not rely on a current trace to maintain its fixed position. This is convenient when comparing two different measurement conditions. To activate a fixed marker on the analyzer, press **Marker, MKR ZERO**.

Marker zero puts a fixed reference at the current position of the active marker.

To change to a Delta Marker to a fixed reference marker, press **Marker, Δ MODE MENU, ΔREF=ΔFIXED MKR**.

Using the MKR ZERO Key to Activate a Fixed Reference Marker

Marker zero enters the position of the active marker as the Δ reference position.

Alternatively, you can specify the fixed point with **FIXED MKR POSITION**. Marker zero is canceled by switching delta mode off.

1. To place marker 1 at a point that you would like to reference, press:

Marker, and turn the front panel knob, or enter a value from the front panel keypad.

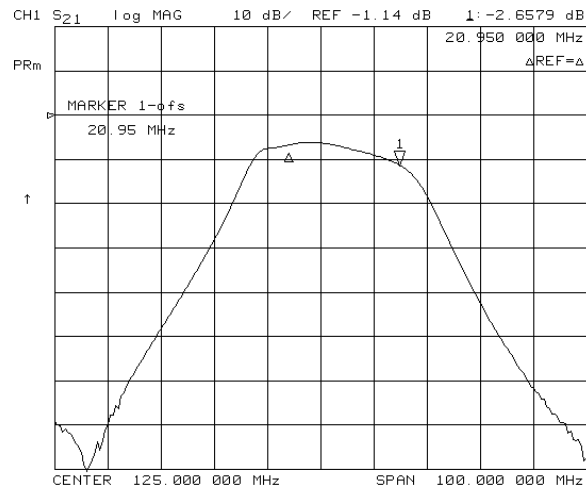
2. To measure values along the measurement data trace, relative to the reference point that you set in the previous step, press:

MKR ZERO, and turn the front panel knob, or enter a value from the front panel keypad.

3. To move the reference position, press:

ΔMODE MENU, FIXED MKR POSITION, FIXED MKR STIMULUS, and turn the front panel knob, or enter a value from the front panel keypad.

Figure 2-12. Example of a Fixed Reference Marker Using MKR ZERO



Using the Δ REF= Δ FIXED MKR, Key to Activate a Fixed Reference Marker

1. To set the frequency value of a fixed marker that appears on the analyzer display, press:

Marker, Δ MODE MENU, Δ REF= Δ FIXED MKR, Δ MODE MENU, FIXED MKR POSITION, FIXED MKR STIMULUS, and turn the front panel knob, or enter a value from the front panel keypad.

The marker is shown on the display as a small delta (Δ), smaller than the inactive marker triangles.

2. To set the response value (dB) of a fixed marker, press:

FIXED MKR VALUE, and turn the front panel knob, or enter a value from the front panel keypad.

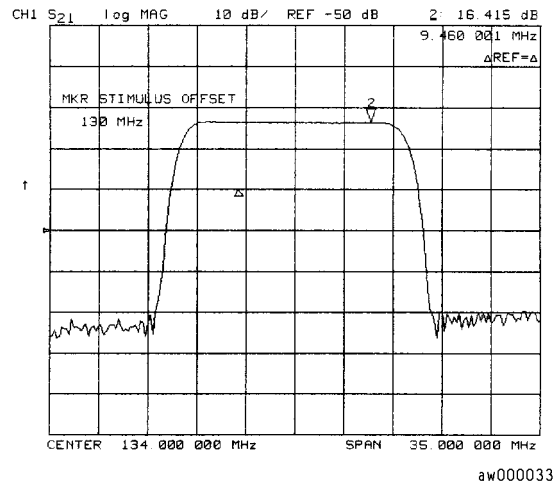
In a Cartesian format, the setting is the y-axis value. In polar or Smith chart format, with a magnitude/phase marker, a real/imaginary marker, an R+jX marker, or a G+jB marker, the setting applies to the first part of the complex data pair. (Fixed marker response values are always uncoupled in the two channels.)

3. To set the auxiliary response value of a fixed marker when you are viewing a polar or Smith format, press:

FIXED MKR AUX VALUE, and turn the front panel knob, or enter a value from the front panel keypad.

This value is the second part of complex data pair, and applies to a magnitude/phase marker, a real/imaginary marker, an R+jX marker, or a G+jB marker. (Fixed marker auxiliary response values are always uncoupled in the two channels.)

Figure 2-13. Example of a Fixed Reference Marker Using $(\Delta)\text{REF}=(\Delta)\text{FIXED MKR}$



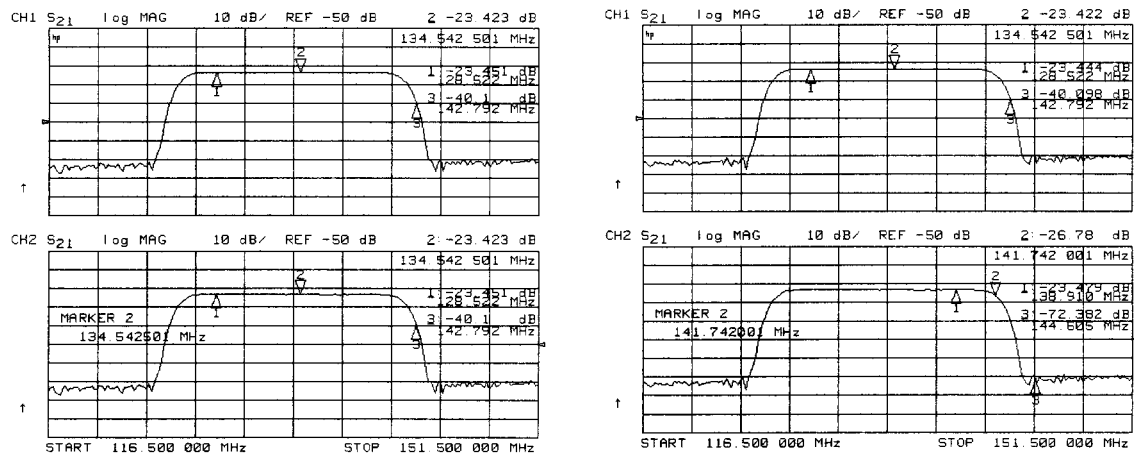
To Couple and Uncouple Display Markers

At a preset state, the markers have the same stimulus values on each channel, but they can be uncoupled so that each channel has independent markers.

Press **Marker Fctn**, **MORE**, **MARKER MODE MENU**, and select from the following keys:

- Choose **MARKERS: COUPLED**, if you want the analyzer to couple the marker stimulus values for the display channels.
- Choose **MARKERS: UNCOUPLED**, if you want the analyzer to uncouple the marker stimulus values for the display channels. This allows you to control the marker stimulus values independently for each channel.

Figure 2-14. Example of Coupled and Uncoupled Markers



pg6235

To Use Polar Format Markers

The analyzer can display the marker value as magnitude and phase, or as a real/imaginary pair: **LIN MKR**, gives linear magnitude and phase, **LOG MKR**, gives log magnitude and phase, **Re/Im**, gives the real value first, then the imaginary value.

You can use these markers only when you are viewing a polar display format. (The format is available from the **Format**, key.)

NOTE For greater accuracy when using markers in the polar format, it is recommended to activate the discrete marker mode. Press **Marker Fctn**, **MORE**, **MKR MODE MENU**, **MARKERS:DISCRETE**.

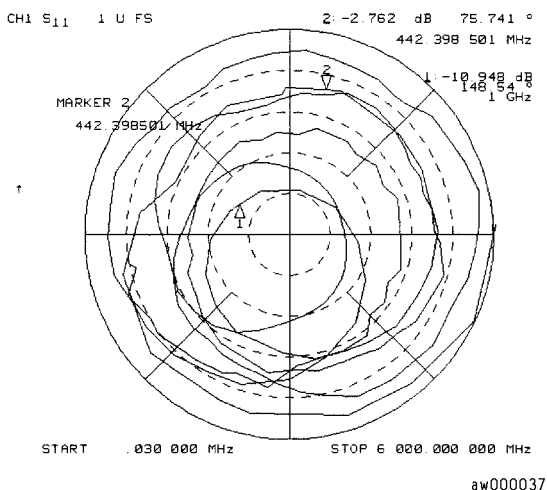
1. To access the polar markers, press:

Format, **POLAR**, **Marker Fctn**, **MORE**, **MARKER MODE MENU**, **POLAR MKR MENU**

2. Select the type of polar marker you want from the following choices:

- Choose **LIN MKR**, if you want to view the magnitude and the phase of the active marker. The magnitude values appear in units and the phase values appear in degrees.
- Choose **LOG MKR**, if you want to view the logarithmic magnitude and the phase of the active marker. The magnitude values appear in dB and the phase values appear in degrees.
- Choose **Re/Im MKR**, if you want to view the real and imaginary pair, where the complex data is separated into its real part and imaginary part. The analyzer shows the real part as the first marker value ($M \cos \Theta$), and the second value is the imaginary part ($M \sin \Theta$, where M = magnitude).

Figure 2-15. Example of a Log Marker in Polar Format



To Use Smith Chart Markers

To avoid displaying marker data interpolated between measured points when using markers in the Smith chart format, activate the discrete marker mode. Press **Marker Fctn, MORE, MKR MODE MENU, MARKERS:DISCRETE**. (Discrete markers will display only measured data and do not interpolate between points.)

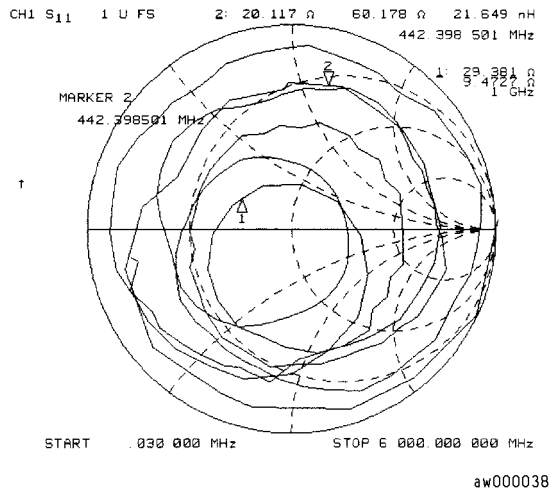
To use Smith chart format:

1. Press **Format, SMITH CHART**.
2. Press **Marker Fctn, MORE, MARKER MODE MENU, SMITH MKR MENU**, and turn the front panel knob, or enter a value from the front panel keypad to read the resistive and reactive components of the complex impedance at any point along the trace. This is the default Smith chart marker.

The marker annotation tells that the complex impedance is capacitive in the bottom half of the Smith chart display and is inductive in the top half of the display.

- Choose **LIN MKR**, if you want the analyzer to show the linear magnitude and the phase of the reflection coefficient at the marker.
- Choose **LOG MKR**, if you want the analyzer to show the logarithmic magnitude and the phase of the reflection coefficient at the active marker. This is useful as a fast method of obtaining a reading of the log magnitude value without changing to log magnitude format.
- Choose **Re/Im MKR**, if you want the analyzer to show the values of the reflection coefficient at the marker as a real and imaginary pair.
- Choose **R+jX MKR**, to show the real and imaginary parts of the device impedance (the series resistance and reactance, in ohms) at the marker. Also shown is the equivalent series inductance or capacitance.
- Choose **G+jB MKR**, to show the complex admittance values of the active marker in rectangular form. The active marker values are displayed in terms of conductance (in Siemens), susceptance, and equivalent parallel circuit capacitance or inductance. Siemens are the international unit of admittance and are equivalent to mhos (the inverse of ohms).

Figure 2-16. Example of Impedance Smith Chart Markers



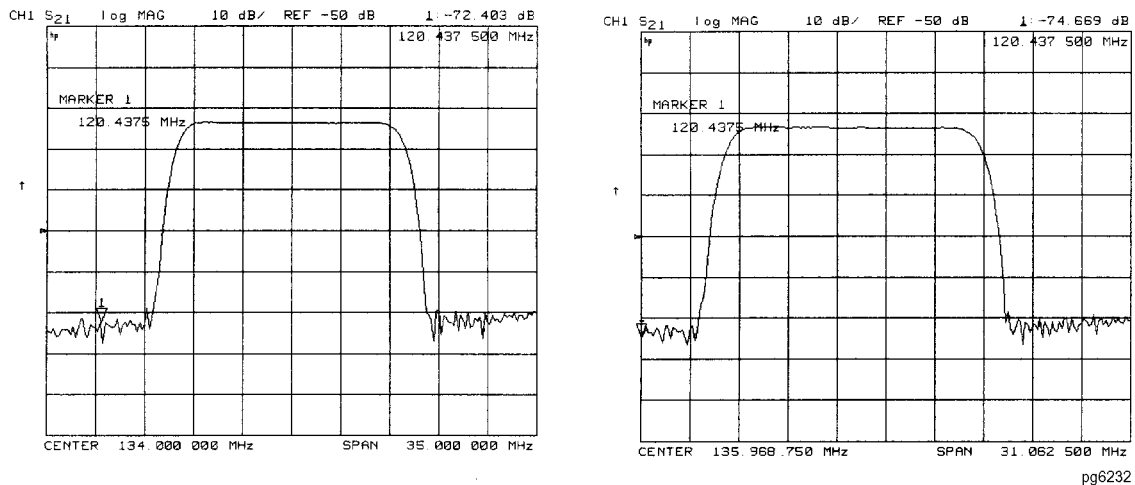
To Set Measurement Parameters Using Markers

The analyzer allows you to set measurement parameters with the markers, without going through the usual key sequence. You can change certain stimulus and response parameters to make them equal to the current active marker value.

Setting the Start Frequency

1. Press **Marker Fctn**, and turn the front panel knob, or enter a value from the front panel keypad to position the marker at the value that you want for the start frequency.
2. Press **MARKER**→**START**, to change the start frequency value to the value of the active marker.

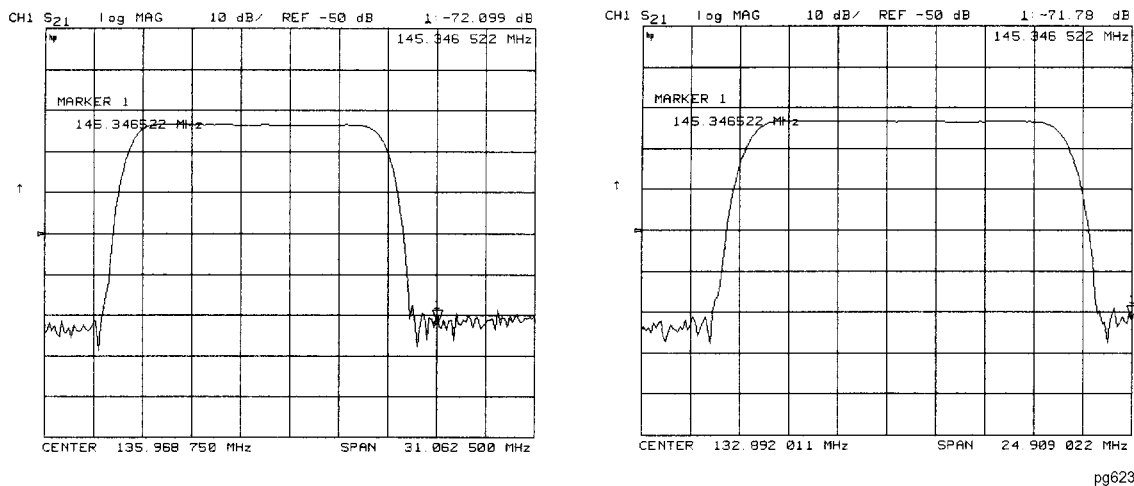
Figure 2-17. Example of Setting the Start Frequency Using a Marker



Setting the Stop Frequency

1. Press **Marker Fctn**, and turn the front panel knob, or enter a value from the front panel keypad to position the marker at the value that you want for the stop frequency.
2. Press **MARKER**→**STOP**, to change the stop frequency value to the value of the active marker.

Figure 2-18. Example of Setting the Stop Frequency Using a Marker

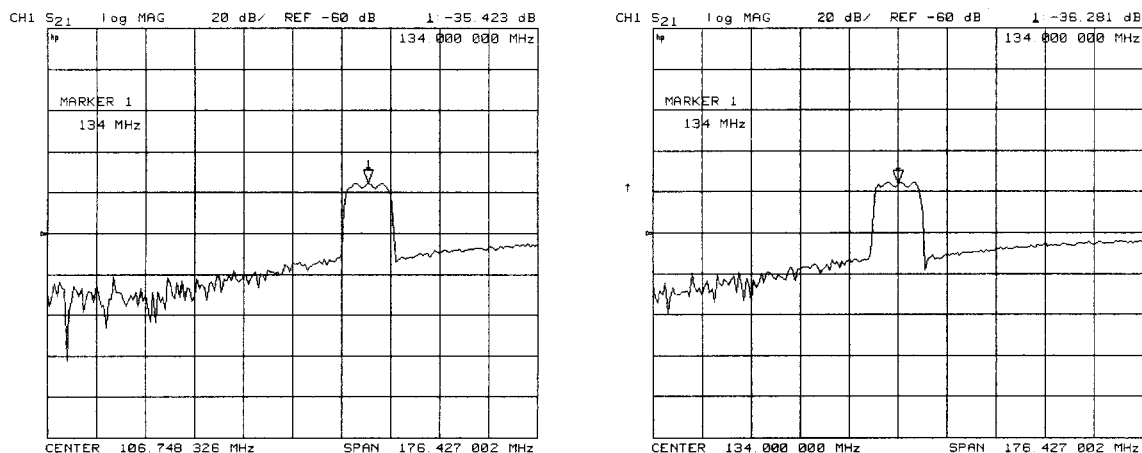


pg6233

Setting the Center Frequency

1. Press **Marker Fctn**, and turn the front panel knob, or enter a value from the front panel keypad to position the marker at the value that you want for the center frequency.
2. Press **MARKER**→**CENTER**, to change the center frequency value to the value of the active marker.

Figure 2-19. Example of Setting the Center Frequency Using a Marker



pg6234

Setting the Frequency Span

You can set the span equal to the spacing between two markers. If you set the center frequency before you set the frequency span, you will have a better view of the area of interest.

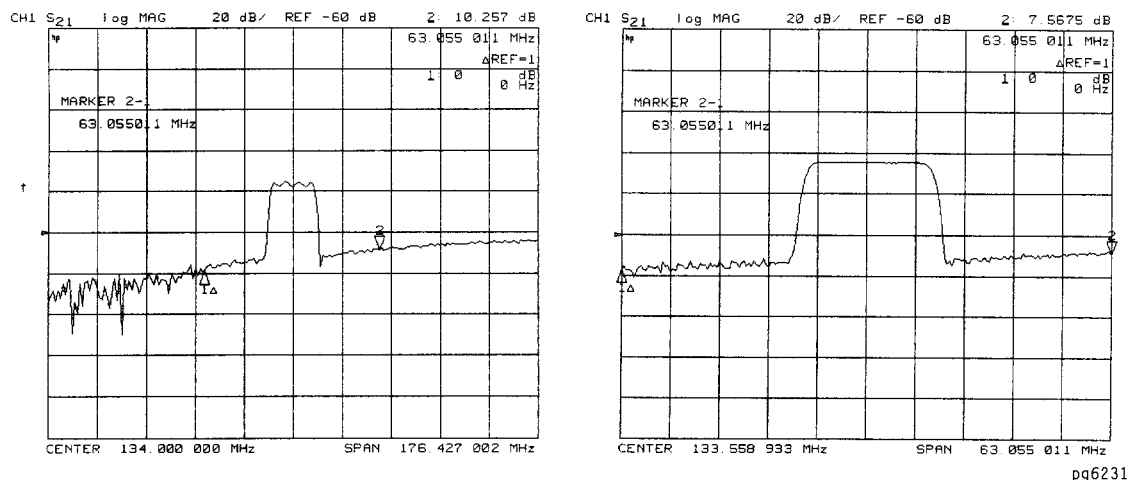
1. Press **Marker**, Δ **MODE MENU**, Δ **REF=1**, **MARKER 2**.
2. Turn the front panel knob, or enter a value from the front panel keypad to position the markers where you want the frequency span.

Iterate between marker 1 and marker 2 by pressing **MARKER 1**, and **MARKER 2**, respectively, and turning the front panel knob or entering values from the front panel keypad to position the markers around the center frequency. When finished positioning the markers, make sure that marker 2 is selected as the active marker.

NOTE Step 2 can also be performed using **MKR ZERO**, and **MARKER 1**. However, when using this method, it will not be possible to iterate between marker zero and marker 1.

3. Press **Marker Fctn**, **MARKER** \rightarrow **SPAN**, to change the frequency span to the range between marker 1 and marker 2.

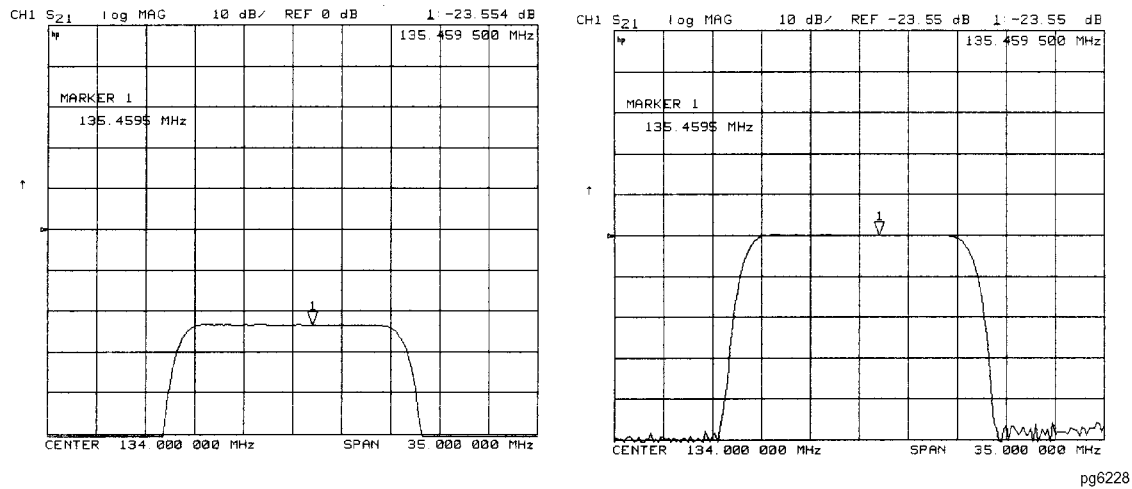
Figure 2-20. Example of Setting the Frequency Span Using Marker



Setting the Display Reference Value

1. Press **Marker Fctn**, and turn the front panel knob, or enter a value from the front panel keypad to position the marker at the value that you want for the analyzer display reference value.
2. Press **MARKER** \rightarrow **REFERENCE**, to change the reference value to the value of the active marker.

Figure 2-21. Example of Setting the Reference Value Using a Marker



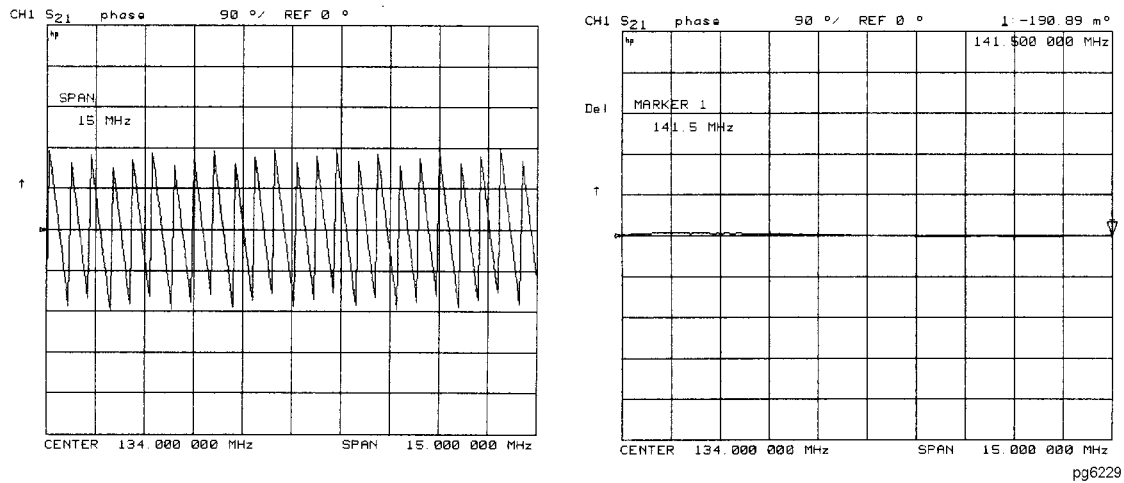
Setting the Electrical Delay

This feature adds phase delay to a variation in phase versus frequency, therefore it is only applicable for ratioed inputs.

1. Press **Format, PHASE**.
2. Press **Marker Fctn**, and turn the front panel knob, or enter a value from the front panel keypad to position the marker at a point of interest.
3. Press **MARKER**→**DELAY**, to automatically add or subtract enough line length to the receiver input to compensate for the phase slope at the active marker position. This effectively flattens the phase trace around the active marker. You can use this to measure the electrical length or deviation from linear phase.

Additional electrical delay adjustments are required on devices without constant group delay over the measured frequency span.

Figure 2-22. Example of Setting the Electrical Delay Using a Marker



Setting the CW Frequency

1. To place a marker at the desired CW frequency, press:

Marker, and either turn the front panel knob or enter the value, followed by a unit terminator.

2. Press **Seq, SPECIAL FUNCTIONS, MKR→CW**.

You can use this function to set the marker to a gain peak in an amplifier. After pressing **MKR→CW FREQ**, activate a CW frequency power sweep to look at the gain compression with increasing input power.

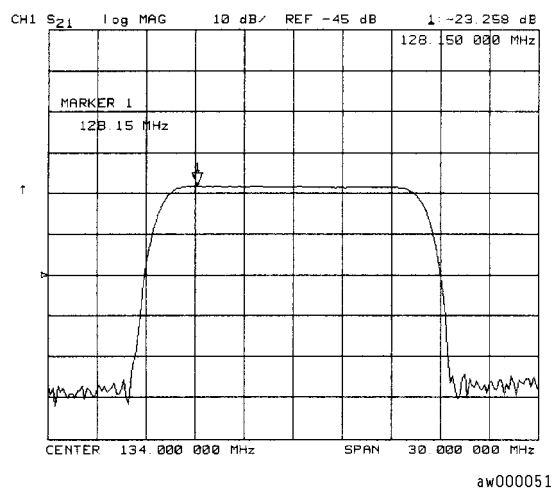
To Search for a Specific Amplitude

These functions place the marker at an amplitude-related point on the trace. If you switch on tracking, the analyzer searches every new trace for the target point.

Searching for the Maximum Amplitude

1. Press **Marker Search**, to access the marker search menu.
2. Press **SEARCH: MAX**, to move the active marker to the maximum point on the measurement trace.

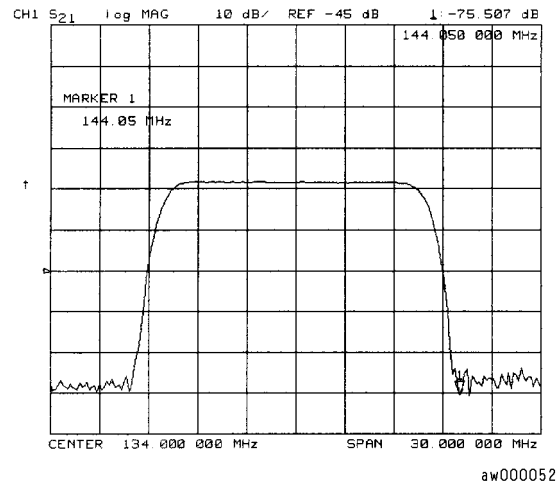
Figure 2-23. Example of Searching for the Maximum Amplitude Using a Marker



Searching for the Minimum Amplitude

1. Press **Marker Search**, to access the marker search menu.
2. Press **SEARCH: MIN**, to move the active marker to the minimum point on the measurement trace.

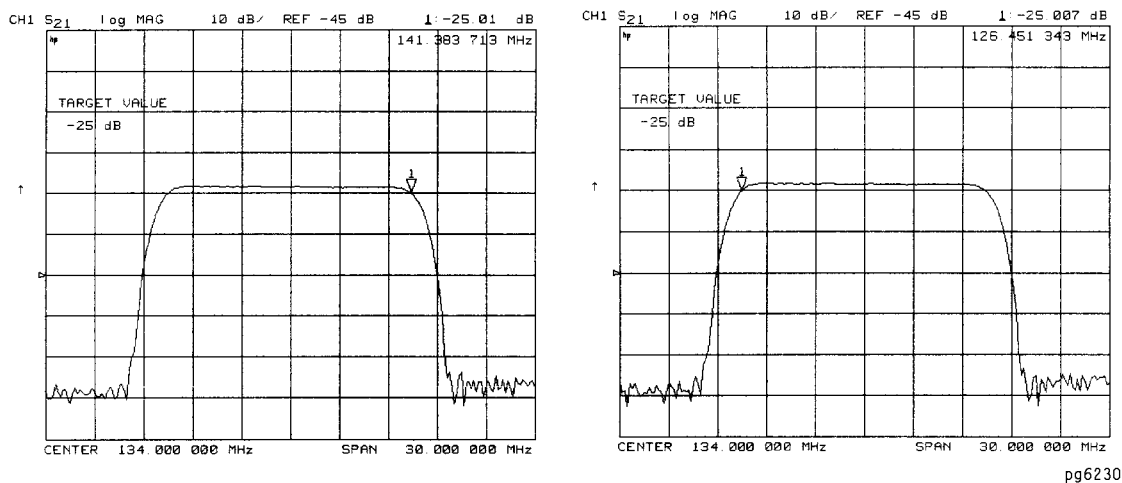
Figure 2-24. Example of Searching for the Minimum Amplitude Using a Marker



Searching for a Target Amplitude

1. Press **Marker Search**, to access the marker search menu.
2. Press **SEARCH: TARGET**, to move the active marker to the target point on the measurement trace.
3. If you want to change the target amplitude value (default is -3 dB), press **TARGET**, and enter the new value from the front panel keypad. You may also press **Marker Search, TARGET VALUE**, to enter the new value.
4. If you want to search for multiple responses at the target amplitude value, press **SEARCH LEFT**, and **SEARCH RIGHT**.

Figure 2-25. Example of Searching for a Target Amplitude Using a Marker



Searching for the 3 dB Bandwidth of a Lowpass Device

The analyzer can search for the 3 dB bandwidth, locating the rolloff on the high side of a device's passband.

1. Press **Marker, Marker 1**.
2. Move marker 1 to any point that you want to reference:

- Turn the front panel knob.

OR

- Enter the frequency value on the numeric keypad.

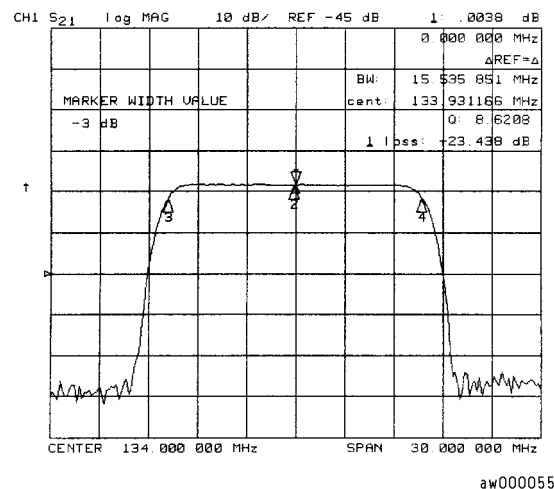
5. Press **Marker Search, 3 DB BANDWIDTH**.

Searching for the Bandwidth and Related Information of a Bandpass Device

The analyzer can automatically calculate and display the bandwidth (BW:), center frequency (CENT:), Q, and loss of the device under test at the center frequency. (Q stands for "quality factor," defined as the ratio of a circuit's resonant frequency to its bandwidth.) These values are shown in the marker data readout.

1. Press **Marker Search**, and **SEARCH: MAX**, to place the marker near the center of the filter passband.
2. Press **Marker, MKR ZERO**, if you want the bandwidth relative to the maximum.
3. Press **Marker Fctn, MORE, BANDWIDTH VALUE** and enter the amplitude value (default is -3 dB) that defines the passband or reject band.
4. If you want to change the amplitude value (default is -3 dB) that defines the passband or reject band, press **WIDTH VALUE**, and enter the new value from the front panel keypad.
5. Press **BANDWIDTH** for the analyzer to calculate the center stimulus value, bandwidth, and the Q of a bandpass or band reject shape on the measurement trace.

Figure 2-26. Example of Searching for the Bandwidth of a Bandpass Device



Tracking the Amplitude that You Are Searching

1. Set up an amplitude search by following one of the previous procedures in “[To Search for a Specific Amplitude](#)” on page 2-32.
2. Press **Marker Search, TRACKING ON**, to track the specified amplitude search with every new trace and put the active marker on that point.

When tracking is not activated, the analyzer finds the specified amplitude on the current sweep and the marker remains at same stimulus value, regardless of changes in the trace response value with subsequent sweeps.

To Calculate the Statistics of the Measurement Data

This function calculates the mean, standard deviation, and peak-to-peak values of the section of the displayed trace between the active marker and the delta reference. If there is no delta reference, the analyzer calculates the statistics for the entire trace.

1. Move marker 1 to any point that you want to reference:
 - Turn the front panel knob.OR
 - Enter the frequency value on the numeric keypad.
2. Press **Marker, Δ MODE MENU, Δ REF=1**, to make marker 1 a reference marker.
3. Press **MARKER 2**, and move marker 2 to any position that you want to measure in reference to marker 1.
4. Press **Marker Fctn, MORE, MKR MODE MENU, MKR STATS ON**, to calculate and view the mean, standard deviation, and peak-to-peak values of the section of the measurement data between the active marker and the delta reference marker.

An application for this feature is to find the peak-to-peak value of passband ripple without searching separately for the maximum and minimum values.

If you are viewing a measurement in the polar or Smith Chart format, the analyzer calculates the statistics using the first value of the complex pair (magnitude, real part, resistance, or conductance).

Figure 2-27. Example Statistics of Measurement Data

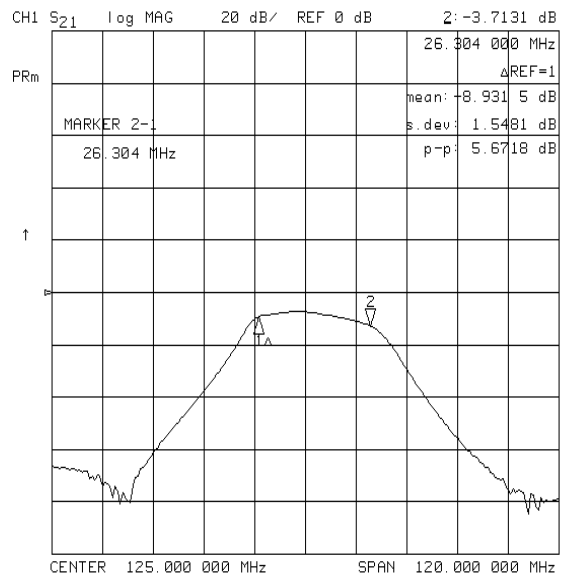


Figure 01

Using the List Mode to Test a Device

When using a list frequency sweep, the analyzer has the ability to sweep arbitrary frequency segments, each containing a list of frequency points. One major advantage of using list frequency sweep is that it allows you to measure the minimum number of data points, and only at the frequencies of interest. This serves to minimize the overall test time. Two different list frequency sweep modes can be selected:

Stepped List

Mode In this mode, the source steps to each defined frequency point, stopping while data is taken. This mode eliminates IF delay and allows frequency segments to overlap. However, the sweep time is substantially slower than for a continuous sweep with the same number of points.

Swept List

Mode *Use this mode for electrical-to-electrical device measurements only.* This mode takes data while sweeping through the defined frequency segments, increasing throughput by up to 6 times over a stepped sweep. In addition, this mode allows the test port power and IF bandwidth to be set independently for each segment that is defined. The frequency segments in this mode cannot overlap.

The ability to completely customize the frequency sweep while using swept list mode is useful when setting up a measurement for a device with high dynamic range, like a filter. The following measurement of a filter illustrates the advantages of using the swept list mode.

- For in-depth information on swept list mode, refer to "[Swept List Frequency Sweep \(Hz\)](#)" on page 7-22.

Connect the Device Under Test

1. Connect the equipment for a transmission measurement.
2. Set the following measurement parameters:

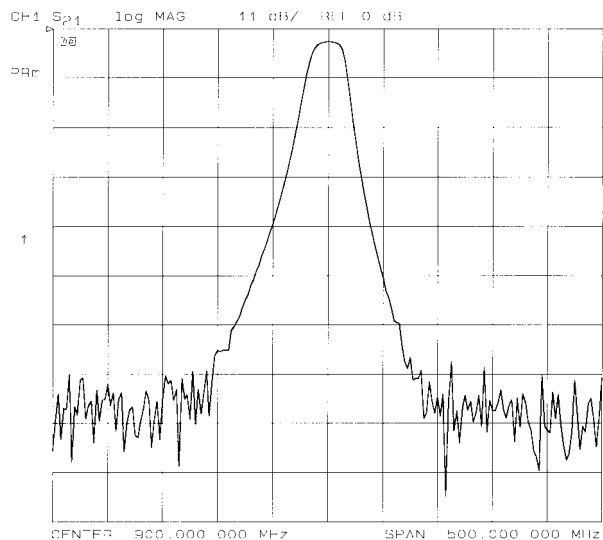
Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)

Center, 900, M/μ

Span, 500, M/μ

Observe the Characteristics of the Filter

Figure 2-28. Characteristics of a Filter



pa5110e

- Generally, the passband of a filter exhibits low loss. A relatively low incident power may be needed to avoid overdriving the next stage of the DUT (if that stage contains an amplifier) or the analyzer receiver.
- Conversely, the stopband of a filter generally exhibits high isolation. To measure this characteristic, the dynamic range of the system will have to be maximized. This can be done by increasing the incident power and narrowing the IF bandwidth.

Choose the Measurement Parameters

1. Decide the frequency ranges of the segments that will cover the stopbands and passband of the filter. For this example, the following ranges will be used:
 - Lower stopband: 650 to 880 MHz
 - Passband: 880 to 920 MHz
 - Upper stopband: 920 to 1150 MHz
2. To set up the swept list measurement, press:
Sweep Setup, SWEEP TYPE MENU, EDIT LIST

Set Up the Lower Stopband Parameters

3. To set up the segment for the lower stopband, press

ADD

START, 650, M/μ

STOP, 880, M/μ

NUMBER of POINTS, 51, x1

4. To maximize the dynamic range in the stopband (increasing the incident power and narrowing the IF bandwidth), press

MORE

LIST POWER ON off, until ON is selected **SEGMENT POWER, 10, x1**

LIST IF BW ON off, until ON is selected **SEGMENT IF BW, 1000, x1**

RETURN, DONE

Set Up the Passband Parameters

5. To set up the segment for the passband, press

ADD

CENTER, 900, M/μ

SPAN, 40, M/μ

STEP SIZE, .2, M/μ

6. To specify a lower power level and a wider IF bandwidth for the passband, press

MORE

SEGMENT POWER, -10, x1

SEGMENT IF BW, 3700, x1

RETURN, DONE

Set Up the Upper Stopband Parameters

7. To set up the segment for the upper stopband, press

ADD

START, 920, M/μ

STOP, 1150, M/μ

NUMBER of POINTS, 51, x1

- To maximize the dynamic range in the stopband (increasing the incident power and narrowing the IF bandwidth), press:

MORE

SEGMENT POWER, 10, x1

SEGMENT IF BW, 300, x1

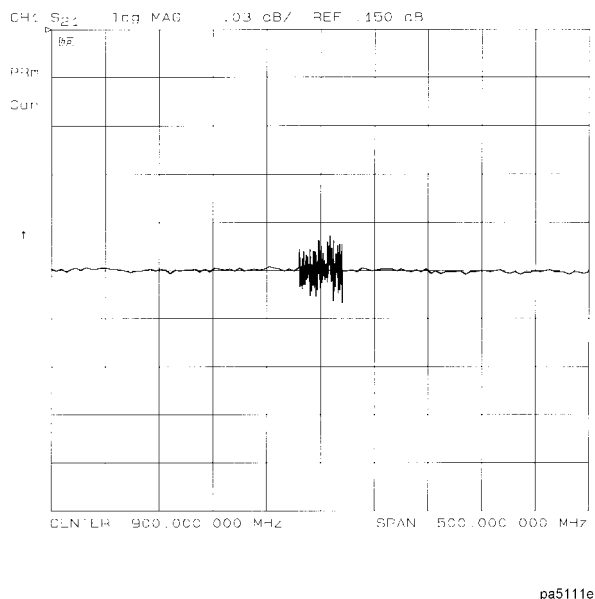
RETURN, DONE

- Press **DONE, LIST FREQ [SWEPT]**.

Calibrate and Measure

- Remove the DUT and perform a full two-port calibration. Refer to [Chapter 4, “Calibrating for Increased Measurement Accuracy”](#)
- With the thru connected, set the scale to autoscale to observe the benefits of using swept list mode.
 - The segments used to measure the stopbands have less noise, thus maximizing dynamic range within the stopband frequencies.
 - The segment used to measure the passband has been set up for faster sweep speed with more measurement points.

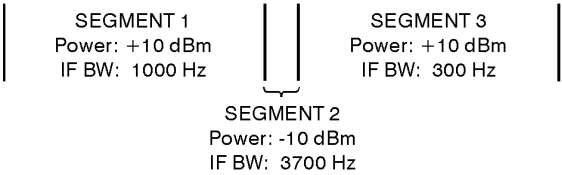
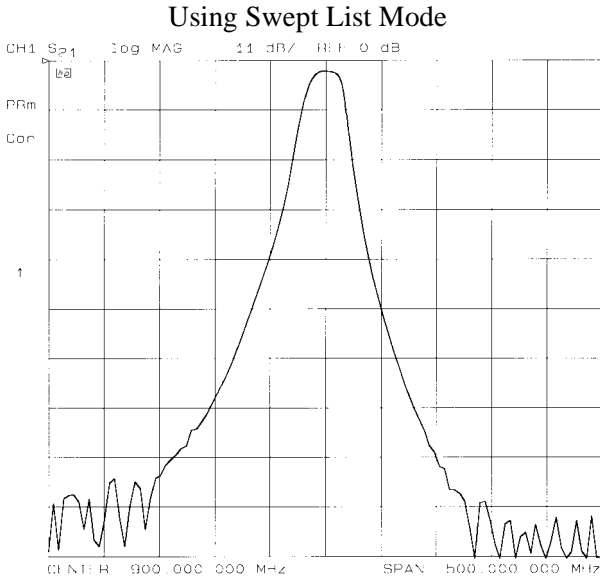
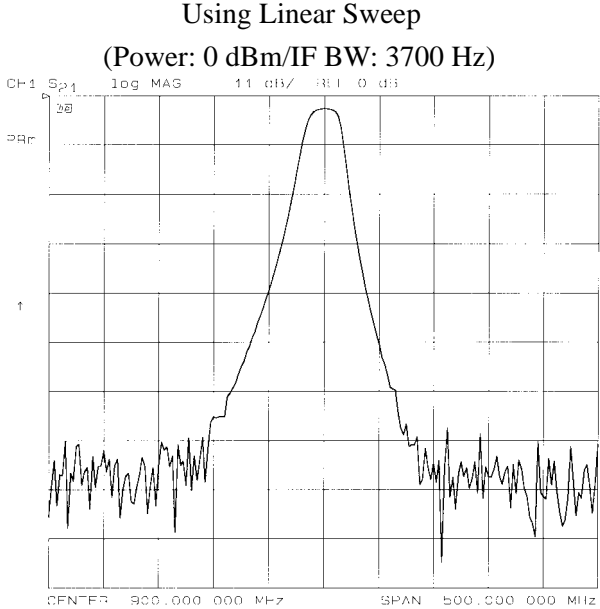
Figure 2-29. Calibrated Swept List Thru Measurement



- Reconnect the filter and adjust the scale to compare results with the first filter measurement that used a linear sweep.

In [Figure 2-29](#), notice that the noise level has decreased over 10 dB, confirming that the noise reduction techniques in the stopbands were successful. Also, notice that the stopband noise in the third segment is slightly lower than in the first segment. This is due to the narrower IF bandwidth of the third segment (300 Hz).

Figure 2-30. Filter Measurements Using Linear Sweep and Swept List Mode



Using Limit Lines to Test a Device

Limit testing is a measurement technique that compares measurement data to constraints that you define. Depending on the results of this comparison, the analyzer will indicate if your device either passes or fails the test.

Limit testing is implemented by creating individual flat, sloping, and single-point limit lines on the analyzer display. When combined, these lines can represent the performance parameters for your device under test. The limit lines created on each measurement channel are independent of each other.

This example measurement shows you how to test a bandpass filter using the following procedures:

- creating flat limit lines
- creating sloping limit lines
- creating single point limit lines
- editing limit segments
- running a limit test

Setting Up the Measurement Parameters

1. Set up the test equipment for a transmission measurement.
2. Press **Preset**, and choose the measurement settings. For this example the measurement settings are as follows:
 - **Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)**
 - **Center, 134, M/μ**
 - **Span, 50, M/μ**
 - **Scale Ref, AUTO SCALE**

You may also want to select settings for the number of data points, power, averaging, and IF bandwidth.

4. Substitute a thru for the device and perform a response calibration by pressing:
Cal, CALIBRATE MENU, RESPONSE, THRU, DONE: RESPONSE
5. Reconnect your test device.

Creating Flat Limit Lines

In this example procedure, the following flat limit line values are set:

Frequency Range	Power Range
127 MHz to 140 MHz	-27 dB to -21 dB
100 MHz to 123 MHz	-200 dB to -65 dB
146 MHz to 160 MHz	-200 dB to -65 dB

NOTE The minimum value for measured data is -200 dB.

1. To access the limits menu and activate the limit lines, press:

System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE, CLEAR LIST, YES

2. To create a new limit line, press:

ADD

The analyzer generates a new segment that appears on the center of the display.

3. To specify the limit's stimulus value, test limits (upper and lower), and the limit type, press:

STIMULUS VALUE, 127, M/ μ

UPPER LIMIT, -21, x1

LOWER LIMIT, -27, x1, DONE

NOTE You could also set the upper and lower limits by using the **MIDDLE VALUE**, and **DELTA LIMITS** keys. To use these keys for the entry, press:
MIDDLE VALUE, -24, x1, DELTA LIMITS, 3, x1

This would correspond to a test specification of -24 ± 3 dB.

4. To define the limit as a flat line, press:

LIMIT TYPE, FLAT LINE, RETURN

5. To terminate the flat line segment by establishing a single point limit, press:

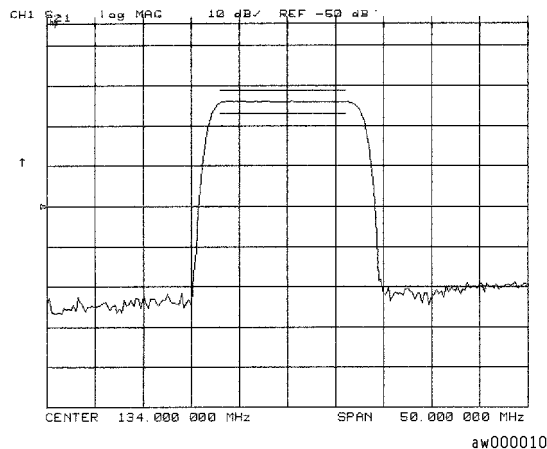
ADD, STIMULUS VALUE, 140, M/ μ , DONE

LIMIT TYPE, SINGLE POINT, RETURN

[Figure 2-31](#) shows the flat limit lines that you have just created with the following parameters:

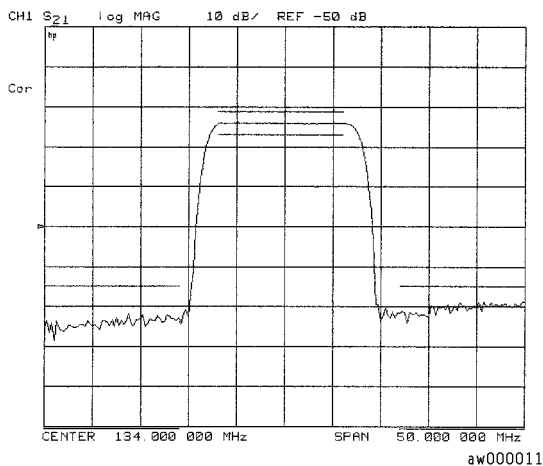
- stimulus from 127 MHz to 140 MHz
- upper limit of -21 dB
- lower limit of -27 dB

Figure 2-31. Example Flat Limit Line



- To create a limit line that tests the low side of the filter, press:
ADD
STIMULUS VALUE, 100, M/μ
UPPER LIMIT, -65, x1
LOWER LIMIT, -200, x1
DONE
LIMIT TYPE, FLAT LINE, RETURN
ADD
STIMULUS VALUE, 123, M/μ
DONE
LIMIT TYPE, SINGLE POINT, RETURN
- To create a limit line that tests the high side of the bandpass filter, press:
ADD
STIMULUS VALUE, 146, M/μ
UPPER LIMIT, -65, x1
LOWER LIMIT, -200, x1
DONE
LIMIT TYPE, FLAT LINE, RETURN
ADD
STIMULUS VALUE, 160, M/μ
DONE
LIMIT TYPE, SINGLE POINT, RETURN

Figure 2-32. Example Flat Limit Lines



Creating a Sloping Limit Line

This example procedure shows you how to make limits that test the shape factor of a SAW filter. The following limits are set:

Frequency Range	Power Range
123 MHz to 125 MHz	-65 dB to -26 dB
144 MHz to 146 MHz	-26 dB to -65 dB

- To access the limits menu and activate the limit lines, press:
System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE, CLEAR LIST, YES
- To establish the start frequency and limits for a sloping limit line that tests the low side of the filter, press:
ADD, STIMULUS VALUE, 123, M/μ
UPPER LIMIT, -65, x1, LOWER LIMIT, -200, x1
DONE, LIMIT TYPE, SLOPING LINE, RETURN
- To terminate the lines and create a sloping limit line, press:
ADD, STIMULUS VALUE, 125, M/μ
UPPER LIMIT, -26, x1, LOWER LIMIT, -200, x1
DONE, LIMIT TYPE, SINGLE POINT, RETURN

4. To establish the start frequency and limits for a sloping limit line that tests the high side of the filter, press:

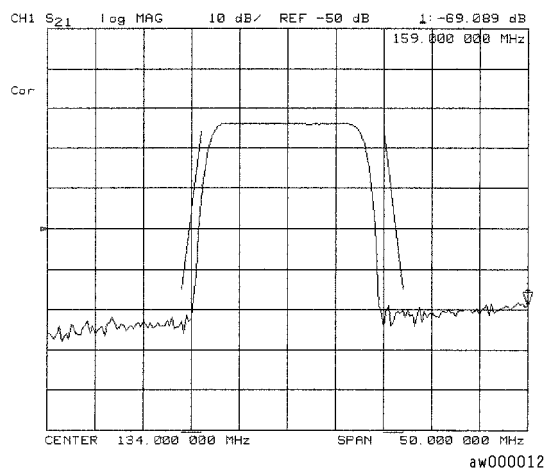
ADD, STIMULUS VALUE, 144, M/ μ
UPPER LIMIT, -26, x1, LOWER LIMIT, -200, x1
DONE, LIMIT TYPE, SLOPING LINE, RETURN

5. To terminate the lines and create a sloping limit line, press:

ADD, STIMULUS VALUE, 146, M/ μ
UPPER LIMIT, -65, x1, LOWER LIMIT, -200, x1
DONE, LIMIT TYPE, SINGLE POINT, RETURN

You could use this type of limit to test the shape factor of a filter.

Figure 2-33. Sloping Limit Lines



Creating Single Point Limits

In this example procedure, the following limits are set:

- from -23 dB to -28.5 dB at 141 MHz
- from -23 dB to -28.5 dB at 126.5 MHz

1. To access the limits menu and activate the limit lines, press:

System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE, CLEAR LIST, YES

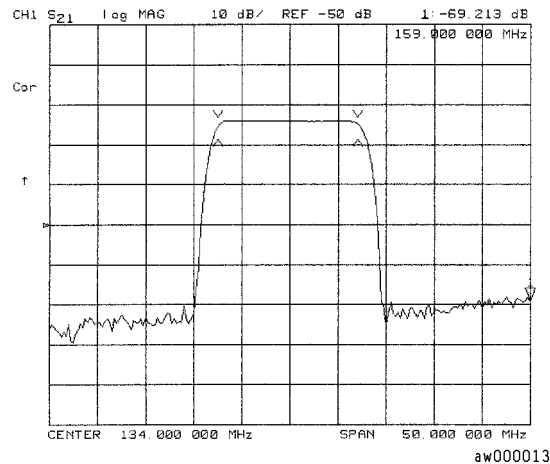
2. To designate a single point limit line, as shown in [Figure 2-34](#), you must define two pointers:

- downward pointing, indicating the upper test limit
- upward pointing, indicating the lower test limit

Press:

```
ADD, STIMULUS VALUE, 141, M/μ
UPPER LIMIT, -23, x1, LOWER LIMIT, -28.5, x1, DONE
LIMIT TYPE, SINGLE POINT, RETURN
ADD, STIMULUS VALUE, 126.5, M/μ
UPPER LIMIT, -23, x1
LOWER LIMIT, -28.5, x1, DONE
LIMIT TYPE, SINGLE POINT, RETURN
```

Figure 2-34. Example Single Points Limit Line



Editing Limit Segments

This example shows you how to edit the upper limit of a limit line.

1. To access the limits menu and activate the limit lines, press:

```
System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE
```

2. To move the pointer symbol (>) on the analyzer display to the segment you wish to modify, press:

```
SEGMENT, up, or down, repeatedly OR SEGMENT, and enter the segment number followed by x1.
```

3. To change the upper limit (for example, -20) of a limit line, press:

```
EDIT, UPPER LIMIT, -20, x1, DONE
```

Deleting Limit Segments

1. To access the limits menu and activate the limit lines, press:
System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE
2. To move the pointer symbol (>) on the analyzer display to the segment you wish to delete, press:
SEGMENT, up, or down, repeatedly OR SEGMENT, and enter the segment number followed by x1.
3. To delete the segment that you have selected with the pointer symbol, press:
DELETE

Running a Limit Test

1. To access the limits menu and activate the limit lines, press:
System, LIMIT MENU, LIMIT LINE, LIMIT LINE ON, EDIT LIMIT LINE

Reviewing the Limit Line Segments

The limit table data that you have previously entered is shown on the analyzer display.

- To verify that each segment in your limits table is correct, review the entries by pressing:
SEGMENT, up, and down
- To modify an incorrect entry, refer to the [“Editing Limit Segments” on page 2-47](#) procedure, located earlier in this section.

Activating the Limit Test

To activate the limit test and the beep fail indicator, press:

System, LIMIT MENU, LIMIT LINE, LIMIT TEST ON, BEEP FAIL ON

NOTE Selecting the beep fail indicator **BEEP FAIL ON**, is optional and will add approximately 50 ms of sweep cycle time. Because the limit test will still work if the limits lines are off, selecting **LIMIT LINE ON**, is also optional.

The limit test results appear on the right side on the analyzer display. The analyzer indicates whether the filter passes or fails the defined limit test:

- The message FAIL will appear on the right side of the display if the limit test fails.
- The analyzer beeps if the limit test fails and if **BEEP FAIL ON**, has been selected.
- The analyzer changes the color of the trace to flashing red where the measurement trace is out of limits.
- A TTL signal on the rear panel BNC connector "LIMIT TEST" provides a pass/fail (5 V/0 V) indication of the limit test results.

Offsetting Limit Lines

The limit offset functions allow you to adjust the limit lines to the frequency and output level of your device. For example, you could apply the stimulus offset feature for testing tunable filters. Or, you could apply the amplitude offset feature for testing variable attenuators, or passband ripple in filters with variable loss.

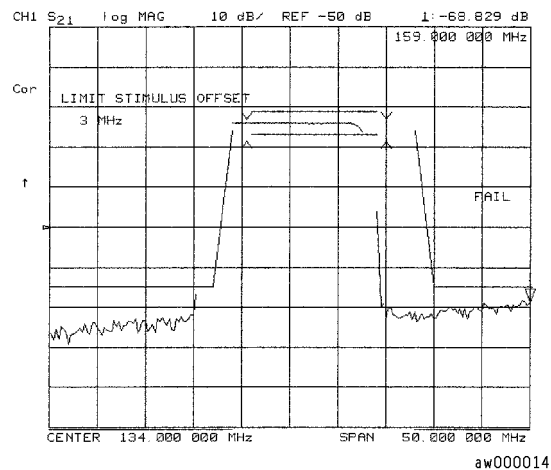
This example shows you the offset feature and the limit test failure indications that can appear on the analyzer display.

- To offset all of the segments in the limit table by a fixed frequency, (for example, 3 MHz), press:

System, LIMIT MENU, LIMIT LINE, LIMIT LINE OFFSETS, STIMULUS OFFSET, 3, M/μ

The analyzer beeps and a FAIL notation appears on the analyzer display, as shown in [Figure 2-35](#).

Figure 2-35. Example Stimulus Offset of Limit Lines



- To return to 0 Hz offset, press:
STIMULUS OFFSET, 0, x1
- To offset all of the segments in the limit table by a fixed amplitude, press:
AMPLITUDE OFFSET, 5, x1
The analyzer beeps and a FAIL notation appears on the analyzer display.
- To return to 0 dB offset, press:
AMPLITUDE OFFSET, 0, x1
- To offset the amplitude offset value by the active marker reading, press **MARKER->AMP.OFS**. Pressing **AMPLITUDE OFFSET**, shows the current value.

Using Ripple Limits to Test a Device

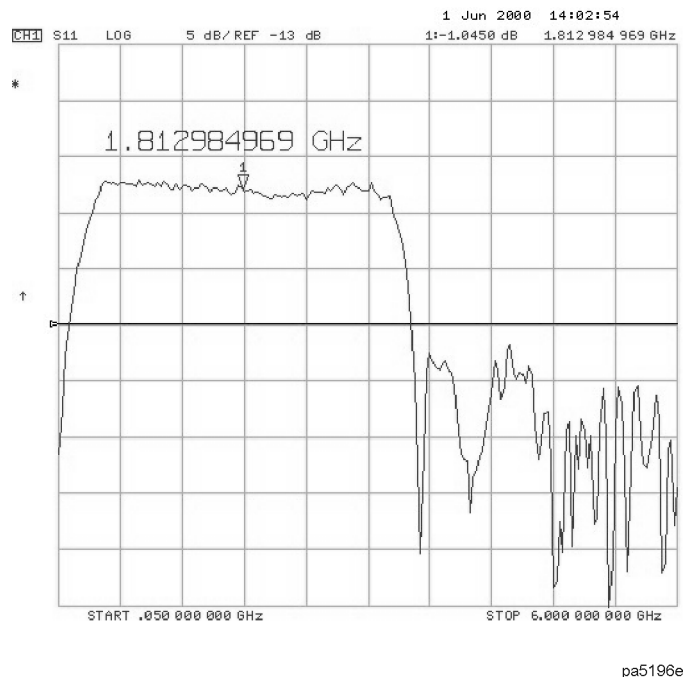
Setting Up the List of Ripple Limits to Test

Two tasks are involved in preparing for ripple testing:

- First, set up the analyzer settings to view the frequency of interest.
- Second, set up the analyzer to test over the appropriate frequencies against your specific limits.

This example will show you how to set up the analyzer to test ripple limits. In this example, we will be testing the pass band of a bandpass filter where the center frequency of the filter is approximately 1.8 GHz and has a bandwidth of approximately 2.9 GHz. Refer to [Figure 2-36](#).

Figure 2-36. Bandpass Filter Being Ripple Tested



Setting Up the Analyzer to Perform the Ripple Test

This section sets up the analyzer so that a bandpass filter can be easily viewed on the analyzer display.

1. Set up the test equipment for a transmission measurement.

2. Press **Preset**, and choose the measurement settings. For this example, the measurement settings are as follows:

- **Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)**
- **Center, 1.8, G/n**
- **Span, 3.4, G/n**
- **Scale Ref, AUTO SCALE**

You may also want to select settings for the number of data points, power, averaging, and IF bandwidth.

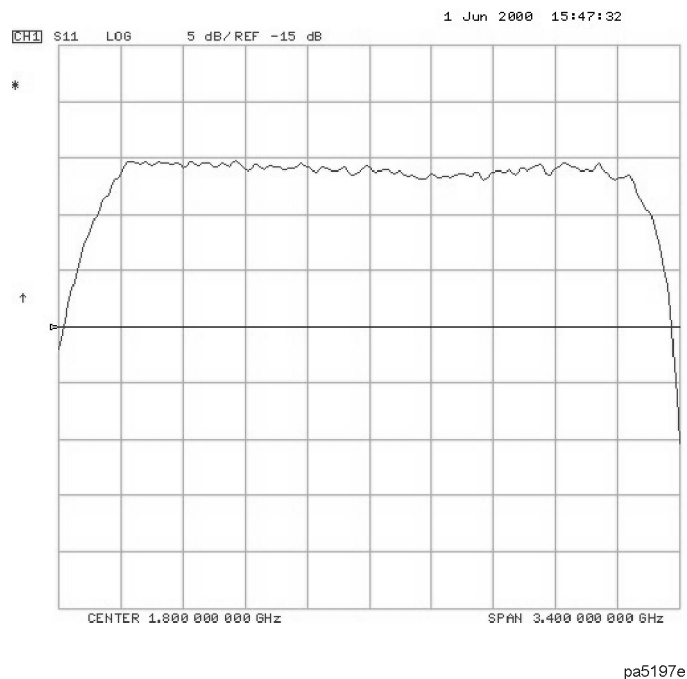
2. Substitute a thru for the device and perform a response calibration by pressing:

Cal, CALIBRATE MENU, RESPONSE, THRU, DONE: RESPONSE

3. Reconnect your test device.

4. To better view the measurement trace, press **Scale Ref, AUTO SCALE**. Refer to [Figure 2-37](#).

Figure 2-37. Filter Pass Band Before Ripple Test



Setting Up Limits for Ripple Testing

This section instructs you on setting up the ripple test parameters. You must set up the analyzer to check the DUT at the correct frequencies and compare the measured values against the maximum allowable ripple value for each frequency band. To do this, you set up individual frequency bands. You define the stop and start frequency and the maximum allowable ripple value of each frequency band. You may set up as many as 12 frequency bands for testing ripple. The frequency bands are combined in a list that is displayed while the ripple frequency bands are being edited.

In this example, we will create one ripple limit (or frequency band) that spans the entire pass

band from 500 MHz to 3.0 GHz. We will also create two additional frequency bands that when merged, will span the pass band with tighter limits. Using the Ripple Edit Menu, we will create a ripple limits list on the analyzer that is similar to [Table 2-5](#).

Table 2-5. Ripple Limits for Ripple Test Example

Frequency Band	Minimum Frequency	Maximum Frequency	Maximum Ripple
1	500 MHz	3.2 GHz	2.0 dB
2	500 MHz	1.85 GHz	1.3 dB
3	1.85 GHz	3.2 GHz	1.3 dB

Notice that Frequency Band 1 overlaps in frequency the remaining frequency bands. Whereas, Frequency Bands 2 and 3 are separate bands that cover the same span of frequency. This can be done to put tighter limits over narrower frequency spans within the bandpass or to customize the ripple test to meet your specific requirements.

1. To access the ripple test menu, press:

System, LIMIT MENU, RIPPLE LIMIT

2. To access the ripple test edit menu, press **EDIT RIPL LIMIT**.

3. Add the first frequency band (Frequency Band 1) to be tested by pressing **ADD**.

4. Set the lower frequency value of Frequency Band 1 by pressing:

MINIMUM FREQUENCY, 500, M/μ

5. Set the upper frequency value of Frequency Band 1 by pressing:

MAXIMUM FREQUENCY, 3.2, G/n

6. Set the maximum allowable ripple amplitude value of Frequency Band 1 by pressing:

MAXIMUM RIPPLE, 2.0, x1

7. Repeat steps 3 through 6 for the two remaining frequency bands to be tested for maximum ripple.

The analyzer allows you to enter up to 12 frequency bands to be tested for maximum ripple.

8. After you have entered all of the ripple test frequency band parameters, return to the ripple test menu by pressing **DONE**.

Editing Ripple Test Limits

Once the frequency band limits for ripple testing have been created, the limits may be changed using the same menu that was used to create them. Using the edit ripple test menu, you may:

- Change existing frequency band limits
- Add more frequency band limits
- Delete individual frequency band limits
- Clear all frequency band limits

Changing Existing Frequency Band Limits

Existing frequency band limits may be changed for testing the ripple. This procedure guides you through changing the existing frequency band limits.

1. To access the ripple test edit menu from the ripple test menu, press:

EDIT RIPL LIMIT

2. Enter the frequency band whose limits you want to change by pressing:

a. FREQUENCY BAND

- b. The numeric key indicating the frequency band number that you are changing.

The frequency band number is located in the left column of the list of frequency bands.

3. Make the changes to the selected band by pressing:

- a. **MINIMUM FREQUENCY**, and the new value to change the lower frequency of the frequency band.

- b. **MAXIMUM FREQUENCY**, and the new value to change the upper frequency of the frequency band.

- c. **MAXIMUM RIPPLE**, and the new decibel value to change the maximum allowable ripple of the frequency band. Terminate the entry of the new decibel value with the **x1**, key.

4. Repeat steps 2 and 3 for additional frequency bands.

5. After you have entered the necessary changes to the ripple test frequency band parameters, return to the ripple test menu by pressing **DONE**.

Adding Additional Frequency Bands

More frequency band limits may be added for testing the ripple. This procedure guides you through adding the more frequency band limits.

The analyzer allows you to enter up to 12 frequency bands for maximum ripple testing.

1. To access the ripple test edit menu, press **EDIT RIPL LIMIT**.
2. Create a new frequency band by pressing **ADD**.

3. Set the lower frequency value of the frequency band by pressing:
 - a. **MINIMUM FREQUENCY**
 - b. the numeric keys indicating the minimum frequency value of the frequency band
 - c. the appropriate frequency key (either **G/n**, **M/μ**, or **k/m**)
4. Set the upper frequency value of the frequency band by pressing:
 - a. **MAXIMUM FREQUENCY**
 - b. the numeric keys indicating the maximum frequency value of the frequency band
 - c. the appropriate frequency key (either **G/n**, **M/μ**, or **k/m**)
5. Set the maximum allowable ripple amplitude value of the frequency band by pressing:
 - a. **MAXIMUM RIPPLE**
 - b. The numeric keys indicating the decibel value of the frequency band's maximum allowable ripple
 - c. **x1**
6. Repeat steps 2 through 5 for additional frequency bands to be tested for maximum ripple.
7. After you have added all of the new frequency bands, return to the ripple test menu by pressing **DONE**.

Deleting Existing Frequency Bands

Frequency band limits may be deleted for testing the ripple. This procedure guides you through deleting existing frequency band limits. You may delete individual frequency bands or delete all of the frequency bands from the list.

1. To access the ripple test edit menu, press:
EDIT RIPL LIMIT
2. Select the first frequency band (as an example, Frequency Band 3) to be deleted by pressing:
FREQUENCY BAND, 3, x1, DELETE
3. Repeat step 2 until you have deleted the required frequency bands from the list.
4. If you need to delete all of the frequency bands, you can delete them all by pressing:
CLEAR LIST
When this softkey is pressed, you will be asked to confirm that you want to delete all of the frequency bands from the list.
5. After you have finished deleting the frequency bands, you can return to the ripple test menu by pressing **DONE**.

Running the Ripple Test

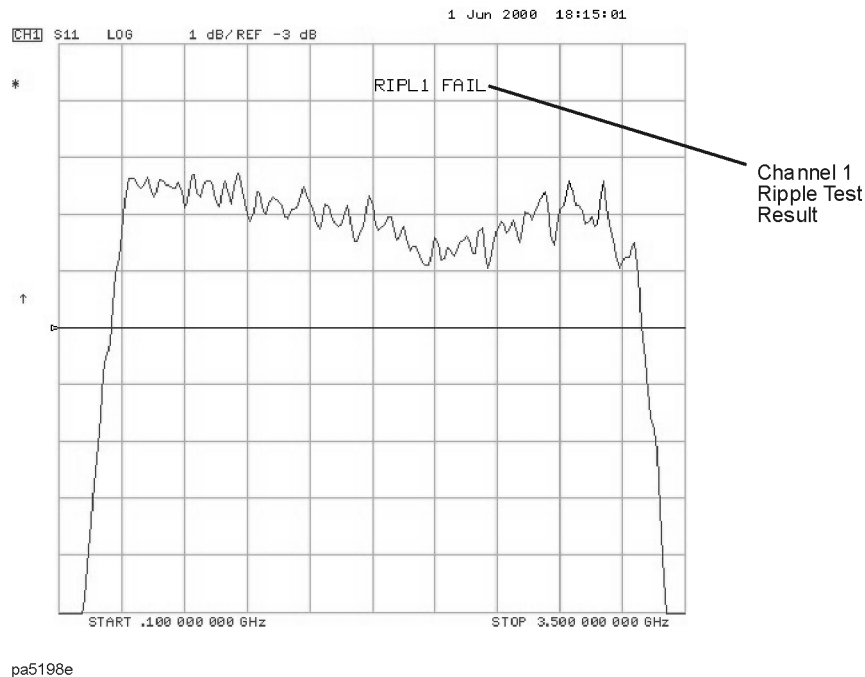
Once the list of ripple limits has been set up, you are ready to run the ripple test. From the Ripple Test Menu, you can:

- Start and stop the ripple test.
- Display and hide the ripple test limit lines.
- Select a frequency band and display its ripple measurement in two ways:
 - the absolute measured ripple value
 - the margin which the measured ripple passes or fails the user-defined maximum ripple value

Starting and Stopping the Ripple Test

Once the list of ripple limits has been set up, start the ripple test by pressing **RIPL TEST on OFF**, from the Ripple Test Menu until **ON** is displayed on the softkey. Pressing this softkey toggles the analyzer between ripple test on and ripple test off status. [Figure 2-38](#) shows the filter pass band (with the scale changed to 1 dB/division) being ripple tested. Note that the filter fails the ripple test. The portions of the pass band trace which do not meet the test requirements are displayed in red.

Figure 2-38. Filter Passband with Ripple Test Activated



As the analyzer measures the ripple, a message is displayed indicating whether the entire measurement passes or fails:

- If the device passes the ripple test, a **RIPL n PASS** message (where n = the channel number) is displayed in the color assigned to the Channel Memory. The device must pass the ripple test in *all* frequency bands before the pass message is displayed.
- If the device fails the ripple test, a **RIPL n FAIL** message (where n = the channel number) is displayed in red. The portion of the trace that exceeds the user-specified maximum ripple value is also displayed in red.
- If the ripple value is not OFF, the test status message also shows you the bands measured and the results:
 - Absolute ripple value selection shows the ripple as an absolute value.
 - Margin ripple value selection shows the ripple in reference to the limits set.

[“Checking the Ripple Value” on page 2-57](#)

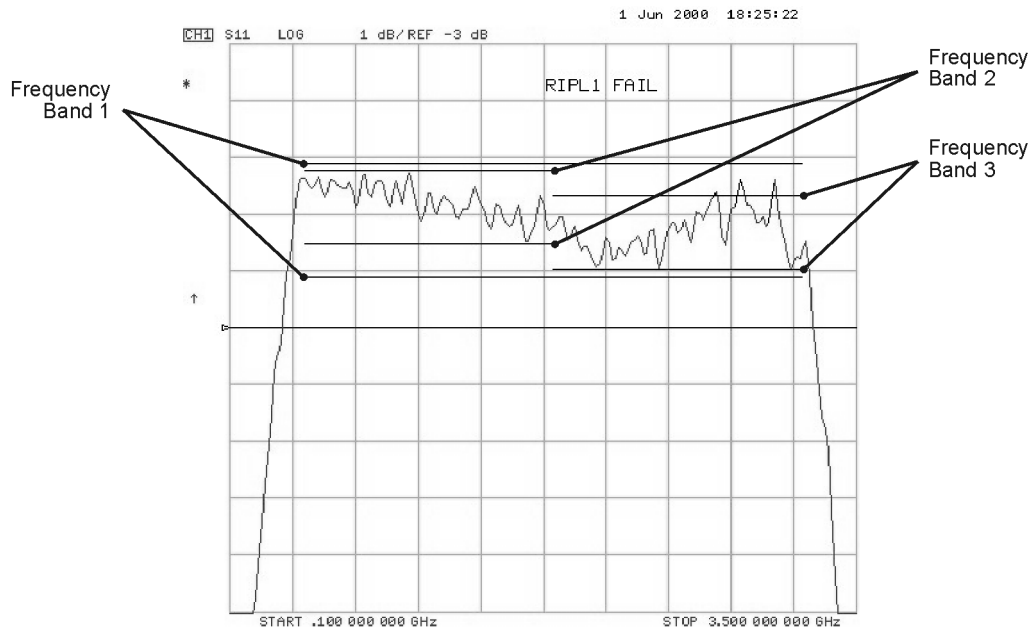
Displaying the Ripple Limits

After the list of ripple limits has been set up, display the ripple test limits by pressing **RIPL LIMIT on OFF**, from the Ripple Test Menu until ON is displayed on the softkey. Pressing this softkey toggles the analyzer ripple limits display on and off. If the ripple limits are displayed and the ripple test is off, the ripple limits are displayed near the top of the graticule and are not compared with the displayed trace. However, once the ripple test is started, the ripple limits are displayed with respect to the measured trace in the following manner:

- If the ripple test passes, the ripple limits are drawn on the display for each frequency band. Within each frequency band, an upper and lower ripple limit is drawn such that they are equidistant above the upper point of the measured trace and below the lower point of the measured trace.
- If the ripple test fails, the ripple limits are drawn on the display for each frequency band. Within each frequency band, the lower ripple limit is drawn at the lowest point on the measured trace and the upper ripple limit is drawn at the user-specified maximum ripple value above the lower ripple limit. The ripple that exceeds the maximum ripple value extends above the upper limit. This measured trace that extends above the upper limit is displayed in red.

[Figure 2-39](#) shows the filter pass band tested with the ripple limits activated. Notice that there are three sets of ripple limits shown. Also notice that the measured trace exceeds the upper ripple limit only in Frequency Band 3.

Figure 2-39. Filter Pass Band with Ripple Test and Ripple Limits Activated



pa5199e

Changing the Ripple Limits Line Color. The color of the lines that represent the ripple limits can be changed by:

- pressing the **Display**, key
- pressing **MORE, ADJUST DISPLAY, MODIFY COLORS, MORE**
- pressing **RIPL LIM LINES, TINT**, and turning the analyzer front panel knob until the desired color appears (You may also use the step keys or the numeric keypad instead of the front panel knob to change the color.)

Checking the Ripple Value

Once the ripple test has been started and is running, you may display the ripple value of each frequency band in one of two formats, the *absolute* format or the *margin* format. Both formats are described in this section.

To display the ripple value, press **RIPL VALUE []**. Pressing this softkey toggles between **RIPL VALUE [OFF]**, **RIPL VALUE [ABSOLUTE]**, and **RIPL VALUE [MARGIN]**. **RIPL TEST on OFF**, from the Ripple Test Menu until ON is displayed on the softkey. Pressing this softkey toggles the analyzer between ripple test on and ripple test off status.

When the Absolute and Margin choices are selected, the frequency band and measurement value are displayed to the right side of the pass/fail message described previously. This portion of the test status message is displayed in the same color as the pass/fail message.

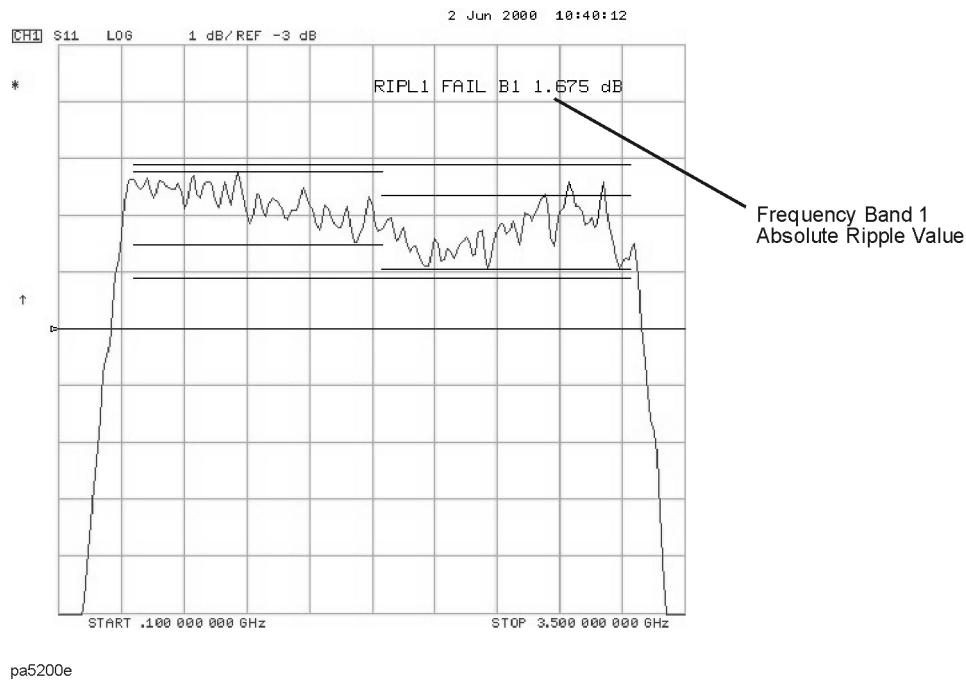
The frequency band of the displayed value is displayed as **B_n** (where **n** = the frequency band number). The frequency band may be changed to display the value of each band. To change the displayed frequency band value, from the Ripple Test Menu press **RIPL VALUE BAND**, and then use the **up**, and **down**, keys (or the numerical keypad) to select the desired frequency band.

Viewing the Ripple Value in Absolute Format

When **RIPL VALUE [ABSOLUTE]** is selected, the absolute ripple value of the selected frequency band is displayed. The absolute ripple value is the measured maximum level minus the measured minimum level within the frequency band. This value is displayed in dB.

Figure 2-40 shows the ripple test with absolute ripple value displayed for Frequency Band 1. The **B1** indicates that the ripple value displayed is for Frequency Band 1. Notice that Frequency Band 1 passes the ripple test. It has an absolute ripple value of 1.675 dB while the maximum ripple value entered for Frequency Band 1 was 2.0 dB. Thus, even though the ripple test fails because of Frequency Band 3, the ripple passes in Frequency Band 1.

Figure 2-40. Filter Pass Band with Absolute Ripple Value for Band 1 Activated

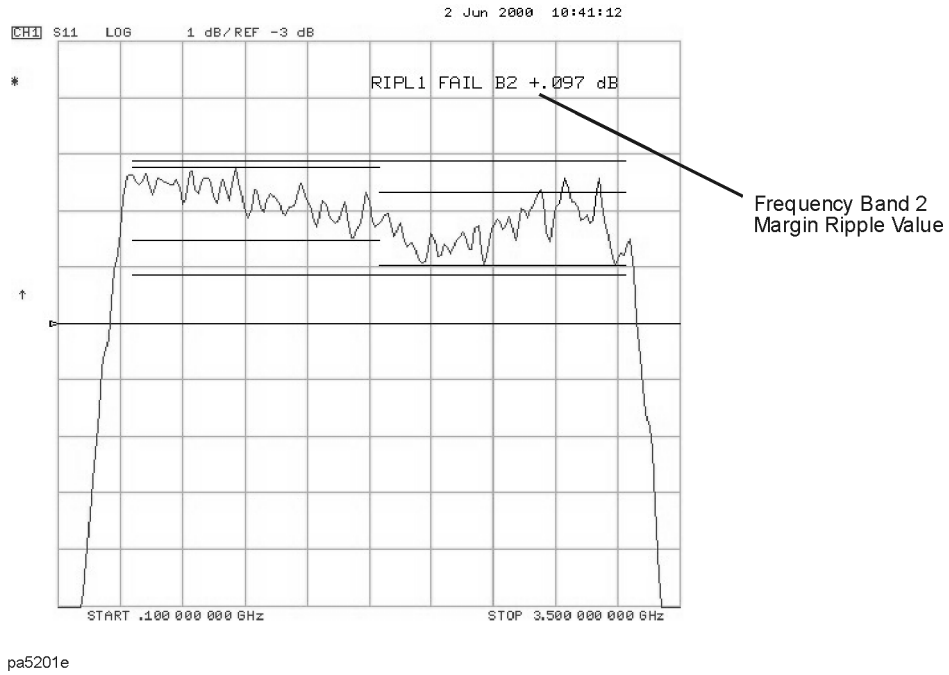


Viewing the Ripple Value in Margin Format

When **RIPL VALUE [MARGIN]**, is selected, the margin by which the ripple value passed or failed is displayed. The ripple value margin is the user-defined maximum ripple minus the absolute ripple value within the frequency band. This value is displayed in dB. A positive value is the margin by which the ripple passes the ripple test in the frequency band. A negative value is the margin by which the ripple fails the ripple test in the frequency band.

Figure 2-41 shows the ripple test with margin ripple value displayed for Frequency Band 2. Notice that Frequency Band 2 passes the ripple test with a margin of 0.097 dB. The plus sign (+) indicates this band passes the ripple test by the amount displayed. A minus sign (-) would indicate that the band failed by the displayed amount.

Figure 2-41. Filter Pass Band with Margin Ripple Value for Band 2 Activated

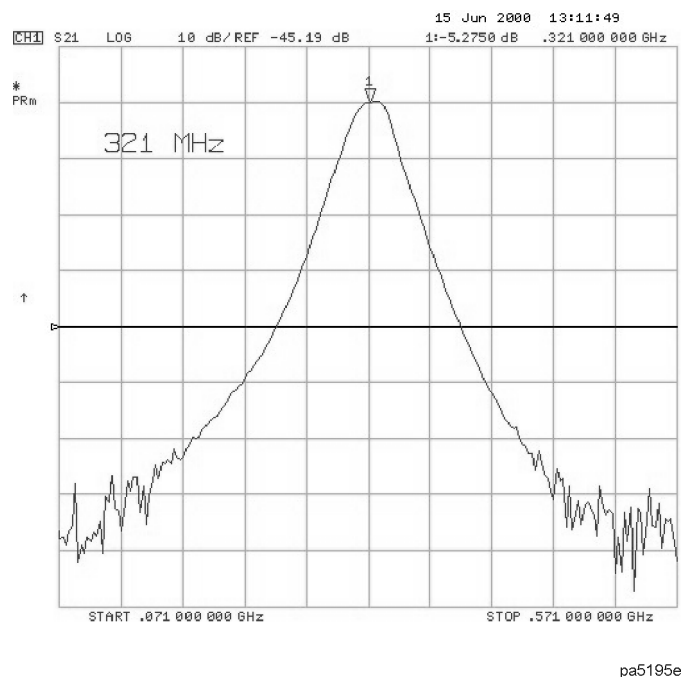


Using Bandwidth Limits to Test a Bandpass Filter

The bandwidth testing mode can be used to test the bandwidth of a bandpass filter. The bandwidth test finds the peak of a signal in the passband and locates a point on each side of the passband at an amplitude below the peak (that you specify during the test setup). The frequency between these two points is the bandwidth of the filter. This bandwidth is compared to minimum and maximum allowable bandwidths that you specify during the test setup.

This example shows you how to test the bandwidth of a bandpass filter. In this example, we will be testing the pass band of a bandpass filter where the center frequency of the filter is approximately 321 MHz. Refer to [Figure 2-42](#).

Figure 2-42. Bandpass Filter Being Bandwidth Tested



Setting Up Bandwidth Limits

When you set up the bandwidth limits to test the bandpass filter, you will first set up the analyzer to perform the bandwidth test and then you will set up bandwidth limits of the bandwidth test.

Setting Up the Analyzer to Perform the Bandwidth Test

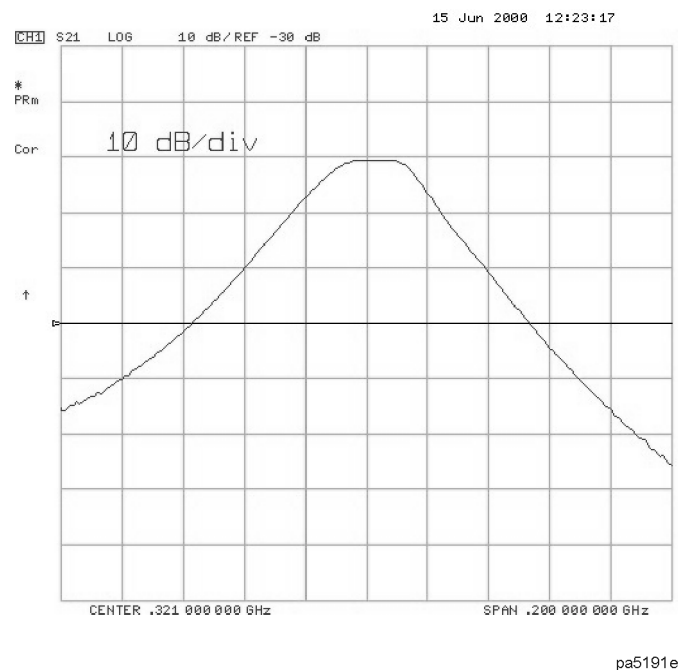
This section sets up the analyzer so that a bandpass filter can be easily viewed on the analyzer display.

1. Set up the test equipment for a transmission measurement.

2. Press **Preset** and choose the measurement settings. For this example, the measurement settings are as follows:
 - a. **Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)**
 - b. **Center, 321, M/ μ**
 - c. **Span, 200, M/ μ**
 - d. **Scale Ref, AUTO SCALE**

You may also want to select settings for the number of data points, power, averaging, and IF bandwidth.

Figure 2-43. Filter Pass Band Before Bandwidth Test



6. Substitute a thru for the device and perform a response calibration by pressing:
Cal, CALIBRATE MENU, RESPONSE, THRU, DONE: RESPONSE
7. Reconnect your test device.

Setting Up the Bandwidth Limits

When you set up the bandwidth limits to test the bandpass filter, you will set

- the *N dB Points* value. The amplitude below the peak that is used to measure the filter's bandwidth.
- the *Maximum Bandwidth* value. If the measured bandwidth is greater than this value, the test will fail.
- the *Minimum Bandwidth* value. If the measured bandwidth is less than this value, the test will fail.

1. To access the bandwidth menu, press:

System, LIMIT MENU, BANDWIDTH LIMIT

2. To set the amplitude below the peak passband amplitude that you want to measure the bandwidth. In this case, we are setting the bandwidth that will be measured 40 dB below the peak amplitude of the bandpass filter by pressing:

N DB POINTS, 40, x1

3. To set the minimum bandwidth for the bandwidth test, press:

MINIMUM BANDWIDTH, 100, M/ μ ,

4. To set the maximum bandwidth for the bandwidth test, press:

MAXIMUM BANDWIDTH, 150, M/ μ

Running a Bandwidth Test

After setting up the bandwidth limits, you are ready to run the bandwidth test and check the test results. For this example, we will:

1. Start the test.
2. Display the bandwidth markers.
3. Review the test results.

Activating the Bandwidth Test

1. Start the bandwidth test by pressing the **BW TEST on OFF** softkey until ON is displayed.

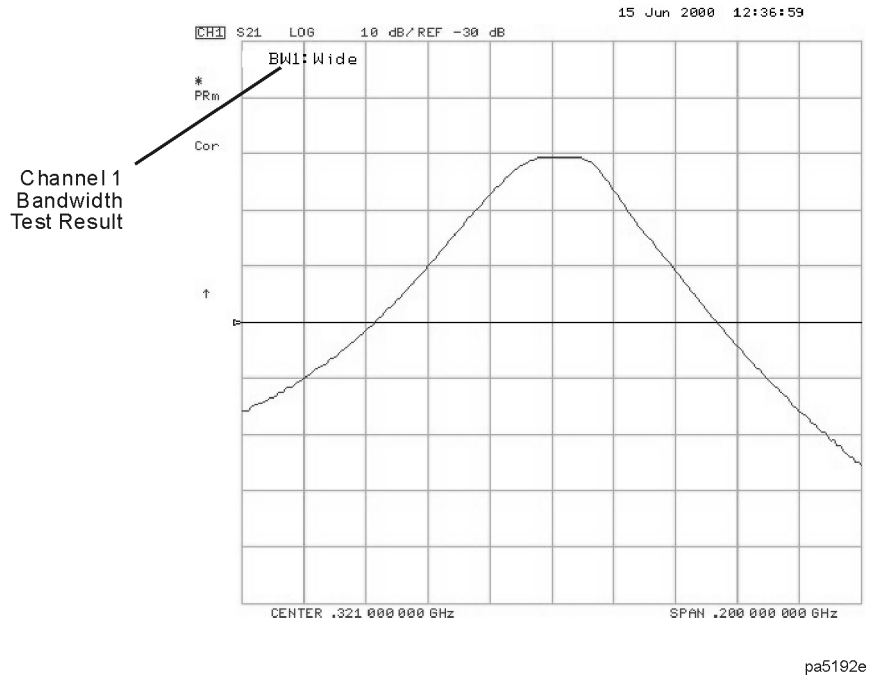
The bandwidth test continues to run until the softkey is returned to the OFF position.

The test displays a message in the upper left corner of the graticule showing that the bandwidth test is being performed and the channel on which the test is being performed. For example, BW1: indicates that the bandwidth test is being run on channel 1. See [Figure 2-44](#).

The test also displays a message indicating whether the filter passes or fails the bandwidth test. When the filter is passing the test, the message indicates Pass. When the filter is failing the test, the failure message indicates either Wide (when the pass band is wider than the maximum bandwidth input) or Narrow (when the pass band is narrower than the minimum bandwidth input).

When the filter passes the bandwidth test, the color of the bandwidth test Pass message is green. When the filter fails the bandwidth test, the color of the bandwidth test Wide/Narrow message is red.

Figure 2-44. Filter Pass Band with Bandwidth Test Activated

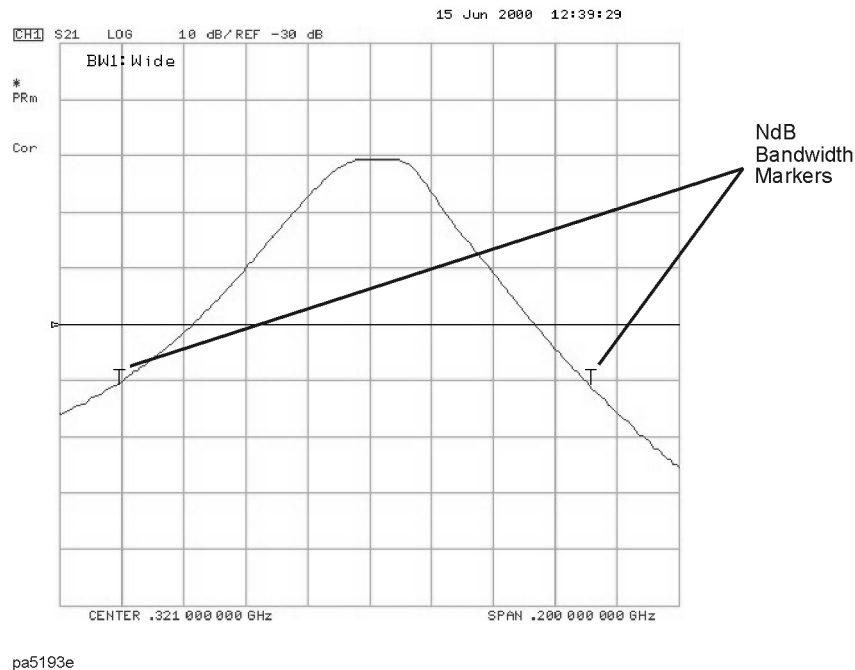


Displaying the Bandwidth Markers

1. Display the bandwidth markers by pressing the **BW MARKER on OFF** softkey until ON is displayed on the softkey.

When the bandwidth markers are displayed, a marker is placed on each side of the peak amplitude at a position equal to the N dB Points value below the peak. The markers are placed at the 40 dB points on the signal in [Figure 2-45](#). The bandwidth markers resemble the following symbol: T

Figure 2-45. Bandwidth Markers Placed 40 dB Below the Bandpass Peak



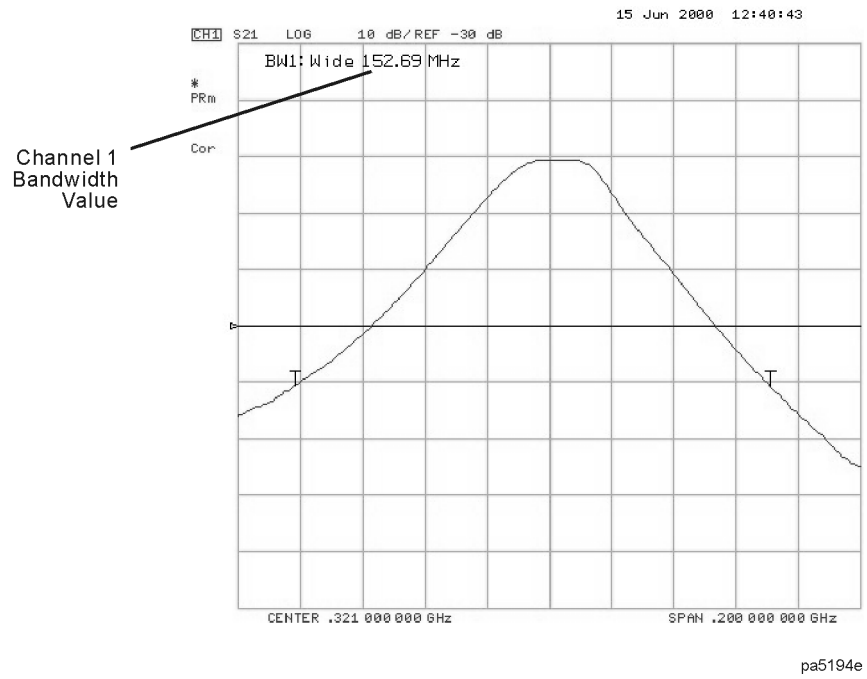
Displaying the Bandwidth Value

1. Display the bandwidth value by pressing the **BW DISPLAY on OFF** softkey until ON is displayed on the softkey.

When this softkey is set to the ON position, the measured bandwidth value is displayed in the upper left corner of the display, to the right of the bandwidth Pass/Wide/Narrow message. This value changes as the analyzer continues measuring the bandwidth. The bandwidth value is displayed in [Figure 2-44](#).

If the filter is failing the bandwidth test, the color of the bandwidth value is red, the same color as the failure (Wide) message of [Figure 2-46](#). If the filter is passing the bandwidth test, the displayed bandwidth value is green (the same color as the bandwidth test Pass message).

Figure 2-46. Filter Pass Band with Bandwidth Value Displayed



Using Test Sequencing

Test sequencing allows you to automate repetitive tasks. As you make a measurement, the analyzer memorizes the keystrokes. Later you can repeat the entire sequence by pressing a single key. Because the sequence is defined with normal measurement keystrokes, you do not need additional programming expertise. Subroutines and limited decision-making increases the flexibility of test sequences. In addition, the GPIO outputs can be controlled in a test sequence, and the GPIO inputs can be tested in a sequence for conditional branching.

The test sequence function allows you to create, title, save, and execute up to six independent sequences internally. You can also save sequences to disk and transfer them from the analyzer to another analyzer or possibly to an external computer controller (so the sequence can be sent to another analyzer).

How to Use Test Sequencing

The following procedures, which are based on an actual measurement example, show you how to do the following:

- create a sequence
- title a sequence
- edit a sequence
- clear a sequence
- change a sequence title
- name files generated by a sequence
- store a sequence
- load a sequence
- purge a sequence
- print a sequence

Creating a Sequence

1. To enter the sequence creation mode, press:

Seq, NEW SEQ/MODIFY SEQ

As shown in [Figure 2-47](#), a list of instructions appear on the analyzer display to help you create or edit a sequence.

Figure 2-47. Test Sequencing Help Instructions

```
TEST SEQUENCING

MODIFY

To INSERT – Any function inserted after cursor.
To DELETE – BACK SP deletes line at cursor.
To STEP – Use ARROW keys or knob. ARROW up does the function after the
           cursor and moves list up. ARROW down only moves list down.
To END – Press DONE SEQ MODIFY in SEQUENCE MENU.

RUN

To START – Press DO SEQUENCE in SEQUENCE MENU.
KEYS – All front panel keys except LOCAL are locked out until
       sequence stops.
To STOP – Press LOCAL to stop a running sequence.
To PAUSE – Press CONTINUE SEQUENCE in SEQUENCE MENU to restart a
           paused sequence.

Only sequence 6 is saved when instrument is turned off.

Select a softkey to start modifying a sequence –
```

2. To select a sequence position in which to store your sequence, press:

SEQUENCE 1 SEQ1

This choice selects sequence position #1. The default title is SEQ1 for this sequence.

Refer to [“Changing the Sequence Title” on page 2-70](#) for information on how to modify a sequence title.

3. To create a test sequence, enter the parameters for the measurement that you wish to make. For this example, a SAW filter measurement is set up with the following parameters:

Save/Recall, SELECT DISK, INTERNAL MEMORY

Use the front panel knob to scroll until **Preset State** is highlighted on the display.

RETURN, RECALL STATE

Meas, ELECTRICAL PARAMETERS, Trans: FWD S21 (B/R)

Format, LOG MAG

Center, 134, M/μ

Span, 50, M/μ

Scale Ref, AUTOSCALE

The previous keystrokes will create a displayed list as shown:

Start of Sequence
RECALL PRST STATE
Trans: FWD S21 (B/R)
LOG MAG
CENTER
134 M/ μ
SPAN
50 M/ μ
SCALE/DIV
AUTO SCALE

4. To complete the sequence creation, press:

Seq, DONE SEQ MODIFY

CAUTION When you create a sequence, the analyzer stores it in volatile memory where it will be lost if you switch off the instrument power (except for sequence #6 which is stored in the analyzer non-volatile memory). However, you may store sequences to a floppy disk.

Running a Sequence

To run a stored test sequence, press:

Preset, and the softkey labeled with desired sequence number.

or, press:

Seq, DO SEQUENCE, and the softkey labeled with the desired sequence number

Stopping a Sequence

To stop a sequence before it has finished, press **Local**.

Editing a Sequence

Deleting Commands

1. To enter the creation/editing mode, press:

Seq, NEW SEQ/MODIFY SEQ

2. To select the particular test sequence you wish to modify (sequence 1 in this example), press:

SEQUENCE 1 SEQ1

3. To move the cursor to the command that you wish to delete, press:

down, or up

- If you wish to scroll through the sequence without executing each line as you do so, you can press the **down** key and scroll through the command list backwards.
- If you use the **up**, key to move the cursor through the list of commands, the commands are actually performed when the cursor points to them. This feature allows the sequence to be tested one command at a time.

4. To delete the selected command, press:

← (backspace key)

5. Press **Seq, DONE SEQ MODIFY** to exit the modify (edit) mode.

Inserting a Command

1. To enter the creation/editing mode, press:

Seq, NEW SEQ/MODIFY SEQ

2. To select the particular test sequence you wish to modify (sequence 1 in this example), press: **SEQUENCE 1 SEQ1**

3. To insert a command, move the cursor to the line immediately above the line where you want to insert a new command, by pressing:

up, or down

- If you use the **up** key to move the cursor through the list of commands, the commands are actually performed when the cursor points to them. This feature allows the sequence to be tested one command at a time.
- If you wish to scroll through the sequence without executing each line as you do so, you can press the **down**, key and scroll through the command list backwards.

4. To enter the new command, press the corresponding analyzer front panel keys. For example, if you want to activate the averaging function, press:

Avg, AVERAGING ON

5. Press **Seq, DONE SEQ MODIFY** to exit the modify (edit) mode.

Modifying a Command

1. To enter the creation/editing mode, press:

Preset, Seq, NEW SEQ/MODIFY SEQ

2. To select the particular test sequence you wish to modify, (sequence 1 in this example), press:

SEQUENCE 1 SEQ1

The following list includes the commands entered in [“Creating a Sequence” on page 2-66](#). Notice that for longer sequences, only a portion of the list can appear on the screen at one time.

```
Start of Sequence
RECALL PRST STATE
Trans: FWD S21 (B/R)
LOG MAG
CENTER
    134 M/μ
SPAN
    50 M/μ
SCALE/DIV
AUTO SCALE
```

3. To change a command (for example, the span value from 50 MHz to 75 MHz), move the cursor (→) next to the command that you wish to modify, press: **up**, or **down**
 - If you use the **up**, key to move the cursor through the list of commands, the commands are actually performed when the cursor points to them. This feature allows the sequence to be tested one command at a time.
 - If you wish to scroll through the sequence without executing each line as you do so, you can press the **down** key and scroll through the command list backwards.
4. To delete the current command (for example, span value), press: ←
5. To insert a new value (for example, 75 MHz), press: **75, M/μ**
6. Press **Seq, DONE SEQ MODIFY** to exit the modify (edit) mode.

Clearing a Sequence from Memory

1. To enter the menu where you can clear a sequence from memory, press: **Seq, MORE, CLEAR SEQUENCE**
2. To clear a sequence, press the softkey of the particular sequence.

Changing the Sequence Title

If you are storing sequences on a disk, you should replace the default titles (SEQ1, SEQ2, ...).

1. To select a sequence that you want to retitle, press: **Seq, MORE, TITLE SEQUENCE** and select the particular sequence softkey.

The analyzer shows the available title characters. The current title is displayed in the upper-left corner of the screen.

2. You can create a new file name in two ways:
 - If you have an attached DIN keyboard, you can press the f6 function key **f6** on the keyboard and type the new file name.
 - If you do not have an attached DIN keyboard, press **ERASE TITLE** and turn the front panel knob to point to the characters of the new file name, pressing **SELECT LETTER** as you stop at each character.

The analyzer cannot accept a title (file name) that is longer than eight characters. Your titles must also begin with a letter, and contain only letters and numbers.

3. To complete the titling, press **DONE**.

Naming Files Generated by a Sequence

The analyzer can automatically increment the name of a file that is generated by a sequence using a loop structure. (See example “[Loop Counter Example Sequence](#)” on page 2-78.)

To access the sequence file name menu, press:

- **Save/Recall**
- **FILE UTILITIES**
- **SEQUENCE FILENAMING**

This menu presents two choices:

- **FILE NAME FILE0** supplies a name for the saved state or data file. This also brings up the Title File Menu.
- **PLOT NAME PLOTFILE** supplies a name for the plot file generated by a plot-to-disk command. This also brings up the Title File Menu.

These keys show the current file name in the 2nd line of the softkey.

When titling a file for use in a loop function, you are restricted to only 2 characters in the file name due to the 6 character length of the loop counter keyword “[LOOP].” When the file is actually written, the [LOOP] keyword is expanded to only 5 ASCII characters (digits), resulting in a 7 character file name.

After entering the 2 character file name, press:

LOOP COUNTER, DONE

Storing a Sequence on a Disk

1. To format a disk, refer to [Chapter 6, “Saving and Recalling Measurements and Data”](#)
2. To save a sequence to the internal disk, press:

Seq, MORE, STORE SEQ TO DISK and select the particular sequence softkey.

The disk drive access light should turn on briefly. When the light goes out, the sequence has been saved.

CAUTION The analyzer will overwrite a file on the disk that has the same title.

CAUTION Do not mistake the line switch for the disk eject button.

Loading a Sequence from Disk

For this procedure to work, the desired file must exist on the disk in the analyzer drive.

1. To view the first six sequences on the disk, press:

Seq, MORE, LOAD SEQ FROM DISK, READ SEQ FILE TITLS

- If the desired sequence is not among the first six files, press:
READ SEQ FILE TITLS until the desired file name appears.

2. Press the softkey next to the title of the desired sequence. The disk access light should illuminate briefly.

NOTE If you know the title of the desired sequence, you can title the sequence (1–6) with the name, and load the sequence. This is also how you can control the sequence number of an imported titled sequence.

Purging a Sequence from Disk

1. To view the contents of the disk (six titles at a time), press:

Seq, MORE, STORE SEQ TO DISK, PURGE SEQUENCES, READ SEQ FILE TITLS

- If the desired sequence is not among the first six files, press:
READ SEQ FILE TITLS until the desired file name appears.

2. Press the softkey next to the title of the desired sequence. The disk access light should illuminate briefly.

Printing a Sequence

1. Configure a compatible printer to the analyzer.
2. To print a sequence, press:

Seq, MORE, PRINT SEQUENCE, and the softkey for the desired sequence.

NOTE If the sequence is on a disk, load the sequence (as described in [“Loading a Sequence from Disk”](#) on page 2-72) and then follow the printing sequence.

In-Depth Sequencing Information

Features That Operate Differently When Executed in a Sequence

The analyzer does not allow you to use the following keys in a sequence:

up, and **down** keys

Preset, key, and

← (backspace key)

Commands That Sequencing Completes Before the Next Sequence Command Begins

The analyzer completes all operations related to the following commands before continuing with another sequence command:

- single sweep
- number of groups
- auto scale
- marker search
- marker function
- data (→) memory
- recall or save (internal or external)
- copy list values and operating parameters
- CHAN1, CHAN2, Wait 0*

*Wait 0 is the special sequencing function **WAIT x**, with a zero entered for the delay value.

Commands That Require a Clean Sweep

Many front panel commands disrupt the sweep in progress, for example, changing the channel or measurement type. When the analyzer does execute a disruptive command in a sequence, some instrument functions are inhibited until a complete sweep is taken. This applies to the following functions:

- autoscale
- data → memory

Forward Stepping in Edit Mode

In the sequence modify mode, you can step through the selected sequence list, where the analyzer executes each step using the **down key**.

Titles

A title may contain non-printable or special ASCII characters if you download it from an external controller. A non-printable character is represented on the display as π .

Sequence Size

A sequence may contain up to 2 kbytes of instructions. Typically, this is around 200 sequence command lines. To estimate a sequence's size (in kbytes), use the following guidelines.

Table 2-6. Guidelines for Determining the Size of a Sequence

Type of Command	Size in Bytes
Typical command	2
Title string character	1
Active entry command	1 per digit

Embedding the Value of the Loop Counter in a Title

You can append a sequentially increasing or decreasing numeric value to the title of stored data by placing a **Display, MORE, TITLE, MORE, LOOP COUNTER** command after the title string. (You must limit the title to three characters if you will use it as a disk file name. The three-character title and five-digit loop counter number reach the eight-character limit for disk file names.) This feature is useful in data logging applications.

Autostarting Sequences

You can define a sequence to run automatically when you apply power to the analyzer. To make an autostarting sequence, create a sequence in position six (SEQ6) and title it "AUTO". To stop an autostarting sequence, press **Local**. To stop an autostarting sequence from engaging at power on, you must clear it from memory or rename it.

NOTE Presetting the instrument does not run the Auto Sequence automatically.

Gosub Sequence Command

The **GOSUB SEQUENCE**, softkey, located in the Sequencing menu, activates a feature that allows the sequence to branch off to another sequence, then return to the original sequence. For example, you could perform an amplifier measurement in the following manner:

1. Create sequence 1 for the specific purpose of performing the gain measurement and printing the results. This sequence will act as a sub-routine.
2. Create sequence 2 to set up a series of different input power levels for the amplifier gain measurements. In-between each power level setting, call sequence 1 as a sub-routine by pressing **GOSUB SEQUENCE, SEQUENCE 1**. Now, sequence 2 will print the measurement results for each input power level applied to the amplifier.

NOTE The **GOSUB SEQUENCE**, softkey branches the sequence to another sequence in a particular location (SEQ1 through SEQ6), not to a given file name.

TTL Out Menu The **TTL OUT** softkey provides access to the TTL out menu. This menu allows you to choose between the following output parameters of the TTL output signal:

- **TTL OUT HIGH**
- **TTL OUT LOW**
- **END SWEEP HIGH PULSE**
- **END SWEEP LOW PULSE**

The TTL output signals are sent to the sequencing BNC rear panel output.

Sequencing Special Functions Menu

This menu is accessed by pressing the **SPECIAL FUNCTIONS**, softkey in the Sequencing menu. This menu provides access to the decision making menu and the more special functions menu. It also contains the peripheral GPIB address and titling, wait x, pause, and marker → CW functions.

Sequence Decision Making Menu

This menu is accessed by pressing the **DECISION MAKING** softkey in the Sequencing Special Functions menu.

Decision making functions are explained in more detail in the following section. These functions check a condition and jump to a specified sequence if the condition is true. The sequence called must be in memory. A sequence call is a one-way jump. A sequence can jump to itself, or to any of the other five sequences currently in memory. Use of these features is explained under the specific softkey descriptions.

Decision Making Functions

Decision making functions jump to a softkey location, not to a specific sequence title

Limit test, loop counter, and do sequence commands jump to any sequence residing in the specified sequence position (1 through 6). These commands do not jump to a specific sequence title. Whatever sequence is in the selected softkey position, will run when these commands are executed.

Having a sequence jump to itself A decision making command can jump to the sequence it is in. When this occurs, the sequence starts over and all commands in the sequence are repeated. This is used a great deal in conjunction with loop counter commands. See the loop counter description that follows.

TTL input decision making TTL input from a peripheral connected to the parallel port (in the GPIO mode) can be used in a decision making function.

Limit test decision making A sequence can jump to another sequence or start over depending on the result of a limit test. When entered into a sequence, the **IF LIMIT TEST PASS**, and **IF LIMIT TEST FAIL** commands require you to enter the destination sequence.

Loop counter decision making The analyzer has a numeric register called a loop counter. The value of this register can be set by a sequence, and it can be incremented or decremented each time a sequence repeats itself. The decision making commands **IF LOOP COUNTER = 0**, and **IF LOOP COUNTER <> 0** jump to another sequence if the stated condition

is true. When entered into the sequence, these commands require you to enter the destination sequence. Either command can jump to another sequence, or restart the current sequence.

As explained in [“Embedding the Value of the Loop Counter in a Title” on page 2-74](#), the loop counter value can be appended to a title. This allows customized titles for data printouts or for data files saved to disk.

Using Test Sequencing to Test a Device

Test sequencing allows you to automate repetitive tasks. As you make a measurement, the analyzer memorizes the keystrokes. Later you can repeat the entire sequence by pressing a single key.

This section contains the following example sequences:

- “Cascading Multiple Example Sequences” on page 2-77
- “Loop Counter Example Sequence” on page 2-78
- “Generating Files in a Loop Counter Example Sequence” on page 2-79
- “Limit Test Example Sequence” on page 2-81

Cascading Multiple Example Sequences

By cascading test sequences, you can create subprograms for a larger test sequence. You can also cascade sequences to extend the length of test sequences to greater than 200 lines.

In this example, you are shown two sequences that have been cascaded. You can do this by having the last command in sequence 1 call sequence position 2, regardless of the sequence title. Because sequences are identified by position, not title, the call operation will always go to the sequence loaded into the given position.

1. To create the example multiple sequences, press:

```
Seq, NEW SEQ/MODIFY SEQ, SEQUENCE 1 SEQ1
Center, 134, M/μ
Span, 50, M/μ
Seq, DO SEQUENCE, SEQUENCE 2
DONE SEQ MODIFY
NEW SEQ/MODIFY SEQ, SEQUENCE 2 SEQ2
Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)
Format, LOG MAG
Scale Ref, AUTOSCALE
Seq, DONE SEQ MODIFY
```

The following sequences will be created:

SEQUENCE SEQ1

Start of Sequence
CENTER
134 M/ μ
SPAN
50 M/ μ
DO SEQUENCE
SEQUENCE 2

SEQUENCE SEQ2

Start of Sequence
Trans:FWD S21 (B/R)
LOG MAG
SCALE/DIV
AUTO SCALE

You can extend this process of calling the next sequence from the last line of the present sequence to 6 internal sequences, or an unlimited number of externally stored sequences.

2. To run both sequences, press:

Preset, SEQUENCE 1 SEQ1

Loop Counter Example Sequence

This example shows you the basic steps necessary for constructing a looping structure within a test sequence. A typical application of this loop counter structure is for repeating a specific measurement as you step through a series of CW frequencies or dc bias levels.

1. To create a sequence that will set the initial value of the loop counter, and call the sequence that you want to repeat, press:

Seq, NEW SEQ/MODIFY SEQ, SEQUENCE 1 SEQ1

SPECIAL FUNCTIONS, DECISION MAKING

LOOP COUNTER, 10, x1

Seq, DO SEQUENCE, SEQUENCE 2

This will create a displayed list as shown:

SEQUENCE LOOP 1

Start of Sequence
LOOP COUNTER
10x1
DO SEQUENCE
SEQUENCE 2

2. Press **DONE SEQ MODIFY**.

3. To create a second sequence that will perform a desired measurement function, decrement the loop counter, and call itself until the loop counter value is equal to zero, press:

```
Seq, NEW SEQ/MODIFY SEQ, SEQUENCE 2 SEQ2
Meas, ELECTRICAL PARAMETERS, Trans:FWD S21 (B/R)
Scale Ref, AUTO SCALE
Marker Search, SEARCH: MAX
Seq, SPECIAL FUNCTIONS, DECISION MAKING
DECR LOOP COUNTER, IF LOOP COUNTER <> 0
SEQUENCE 2 SEQ2
Seq, DONE SEQ MODIFY
```

This will create a displayed list as shown:

```
SEQUENCE LOOP 2
Start of Sequence
Trans:FWD S21 (B/R)
SCALE/DIV
AUTO SCALE
MKR Fctn
SEARCH MAX
DECR LOOP COUNTER
IF LOOP COUNTER <> 0 THEN DO
  SEQUENCE 2
```

To run the loop sequence, press:

Preset, SEQUENCE 1 SEQ1,

Generating Files in a Loop Counter Example Sequence

This example shows how to increment the names of files that are generated by a sequence with a loop structure.

```
Seq,NEW SEQ/MODIFY SEQ, SEQUENCE 1 SEQ 1
SPECIAL FUNCTIONS, DECISION MAKING
LOOP COUNTER, 7, x1
Save/Recall, SELECT DISK, INTERNAL DISK
RETURN, DEFINE DISK-SAVE, DATA ONLY ON
Local, SET ADDRESSES, PLOTTER PORT, DISK
Seq, DO SEQUENCE, SEQUENCE 2
DONE SEQ MODIFY
NEW SEQ/MODIFY SEQ, SEQUENCE 2 SEQ 2
Save/Recall, FILE UTILITIES, SEQUENCE FILE NAMING
FILE NAME FILE0, ERASE TITLE
```

D, T, LOOP COUNTER, DONE
PLOT NAME PLOTFILE,ERASE TITLE
P, L, LOOP COUNTER, DONE, RETURN
Sweep Setup, TRIGGER MENU, SINGLE
Save/Recall, SAVE STATE
Copy, PLOT
Seq, SPECIAL FUNCTIONS, DECISION MAKING
DECR LOOP COUNTER, IF LOOP COUNTER 0
SEQUENCE 2 SEQ 2
Seq, DONE SEQ MODIFY

This will create the following displayed lists:

```
Start of Sequence
LOOP COUNTER
  7 x1
INTERNAL DISK
DATA ONLY
ON
DO SEQUENCE
  SEQUENCE 2
```

```
Start of Sequence
FILE NAME
  DT[LOOP]
PLOT NAME
  PL[LOOP]
SINGLE
SAVE FILE 0
PLOT
DECR LOOP COUNTER
IF LOOP COUNTER <> 0 THEN DO
  SEQUENCE 2
```

Sequence 1 initializes the loop counter and calls sequence 2. Sequence 2 repeats until the loop counter reaches 0. For each loop, it takes a single sweep, saves the data file and plots the display.

- The data file names generated by this sequence will be:

```
DT00007.D1
through
DT000001.D1
```

- The plot file names generated by this sequence will be:

PL00007.FP

through

PL00001.FP

To run the sequence, press:

Preset, SEQUENCE 1 SEQ 1

Limit Test Example Sequence

This measurement example shows you how to create a sequence that will branch the sequence according to the outcome of a limit test. Refer to [“Using Limit Lines to Test a Device” on page 2-42](#), for a procedure that shows you how to create a limit test.

For this example, you must have already saved the following in register 1:

- device measurement parameters
 - a series of active (visible) limit lines
 - an active limit test
1. To create a sequence that will recall the desired instrument state, perform a limit test, and branch to another sequence position based on the outcome of that limit test, press:

Seq, NEW SEQ MODIFY, SEQUENCE 1 SEQ1

Save/Recall, RECALL KEYS, RECALLKEYS MENU, RECALL REG1

Seq, SPECIAL FUNCTIONS, DECISION MAKING

IF LIMIT TEST PASS, SEQUENCE 2 SEQ2

IF LIMIT TEST FAIL, SEQUENCE 3 SEQ3

Seq, DONE SEQ MODIFY

This will create a displayed list for sequence 1, as shown:

```
Start of Sequence
RECALL REG 1
IF LIMIT TEST PASS THEN DO
SEQUENCE 2
IF LIMIT TEST FAIL THEN DO
SEQUENCE 3
```

2. To create a sequence that stores the measurement data for a device that has passed the limit test, press:

Seq, NEW SEQ MODIFY, SEQUENCE 2 SEQ2

Save/Recall, SELECT DISK, INTERNAL DISK, RETURN

DEFINE DISK-SAVE, DATA ARRAY ON, RETURN, SAVE STATE

Seq, DONE SEQ MODIFY

This will create a displayed list for sequence 2, as shown:

```
Start of Sequence
INTERNAL DISK
DATA ARRAY
ON
FILENAME
FILE 0
SAVE FILE
```

3. To create a sequence that prompts you to tune a device that has failed the limit test, and calls sequence 1 to retest the device, press:

```
Seq, NEW SEQ/MODIFY SEQ, SEQUENCE 3 SEQ3
Display, MORE, TITLE
T, U, N, E, D, E, V, I, C, E, DONE
Seq, SPECIAL FUNCTIONS, PAUSE, RETURN
DO SEQUENCE, SEQUENCE 1 SEQ1
Seq, DONE SEQ MODIFY
```

This will create a displayed list for sequence 3, as shown:

```
Start of Sequence
TITLE
TUNE DEVICE
SEQUENCE
PAUSE
DO SEQUENCE
SEQUENCE 1
```

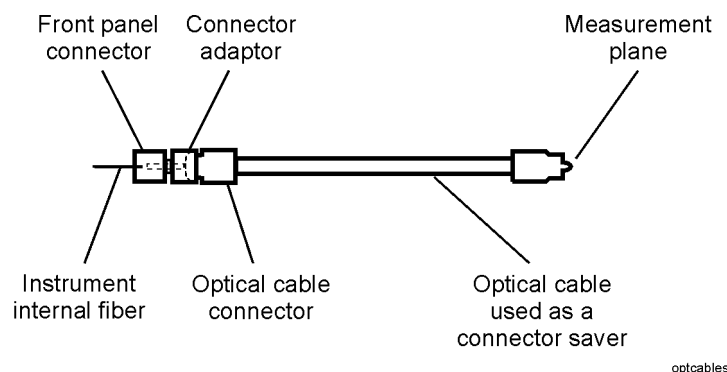
You will see the “tune device” prompt in the title area or the sequence will pause until you press the **CONTINUE SEQUENCE** key.

Magnitude and Phase Comparisons of Lightwave Receivers

The application described in this example is the measurement and comparison of two lightwave receiver responses. The magnitude of the receivers are measured and a data/memory math function is performed on the traces for magnitude comparison. The two responses are then displayed in the phase format and electrical length is added to read the electrical delay of the devices. Again, a math function is performed on the traces for comparison.

This application can be useful when measuring two similar devices and performing alignment adjustment in real time for matching phase and differential characteristics.

CAUTION COSTLY REPLACEMENT of an entire lightwave assembly will result from damage to an optical test port connector. Front panel test port connectors are integral parts of the lightwave components and ARE NOT separately replaceable. Keep optical cables connected to the test ports to protect the connectors from damage.



1. To engage the dual display feature and set up the type of measurement for two channels, press:

Preset, Display, DUAL/QUAD SETUP, DUAL CHAN ON

Meas, Trans: O/E (PORT 2)

Chan 2, Trans: O/E (PORT 2)

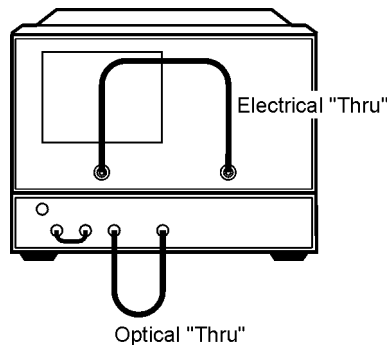
Sweep Setup, SWEEP TIME MANUAL, 5, x1

Avg, IF BW, 100, x1

(The IF bandwidth is decreased to filter out noise and lower the noise floor.)

2. Connect the optical and electrical cables as shown below. Make sure all the connections are clean. Refer to [“Cleaning Connections for Accurate Measurements”](#) on page 1-2 for cleaning information.

Figure 2-48. Response Calibration Setup of an O/E Transmission Measurement



oe_r_m3

3. To perform a response measurement calibration, press:

Cal, CALIBRATE MENU, RESPONSE, THRU

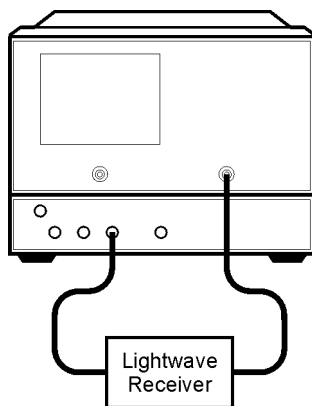
When the calibration measurement is complete, the analyzer underlines the softkey label "THRU". Press **DONE: RESPONSE**.

4. To save the measurement state and calibration in a register, press **Save/Recall, SAVE STATE**.

Magnitude Response

5. Connect the cables to one of the lightwave receivers as shown below. Be sure all the connections are clean.

Figure 2-49. O/E Transmission Measurement Configuration



oe_transm

6. To better view the measurement trace, press **Scale Ref, AUTO SCALE**.
7. To input the data trace of channel 2 (the active channel) to memory 2, press **Display, DATA->MEMORY 2, DISPLAY: MEMORY**.

8. The marker stimulus values are coupled (by default) for the two display channels. Press the following keys to uncouple the markers:

Marker Fctn, MORE, MARKER MODE MENU, MARKERS: UNCOUPLED

When the marker stimulus is uncoupled, and the dual channel features is active, two sets of stimulus values are shown on the analyzer display.

9. To locate the maximum amplitude of the device response press **Marker Search, SEARCH: MAX.**
10. To change the active channel to channel 1 and measure the other lightwave receiver while maintaining the display of the channel 2 measurement trace, press **Chan 1.**
11. Connect the second lightwave receiver to the analyzer as in step 5. Make sure all the connections are clean.
12. To better view the measurement trace, press **Scale Ref, AUTO SCALE.**
13. To locate the maximum amplitude of the device response press **Marker Search, SEARCH: MAX.**

In this example, there is a 1.6483 dB difference between the maximum amplitude values of the two receivers. If the intention is to match the devices as closely as possible to each other, the device measured in channel 1 could have an attenuator placed at the RF output to decrease the output power and match the device power measured on channel 2. Refer to [“Matching Device Output Power” on page 2-86.](#)

14. To compare the two receiver responses by applying the trace math feature, follow these steps:

To view a single channel to more easily see the math results, press **Chan 1, Display, DUAL/QUAD SETUP, DUAL CHAN OFF.**

To bring the trace stored on memory 2 (channel 2) to memory 1 (channel 1) press **RETURN, SELECT DEFAULTS, MEMORY2->MEMORY1.**

To see the ratio of the receiver response measured on channel 1 to the receiver response measured on channel 2, press **RETURN, MATH: DATA/MEM.**

Figure 2-50. Matching Device Output Power

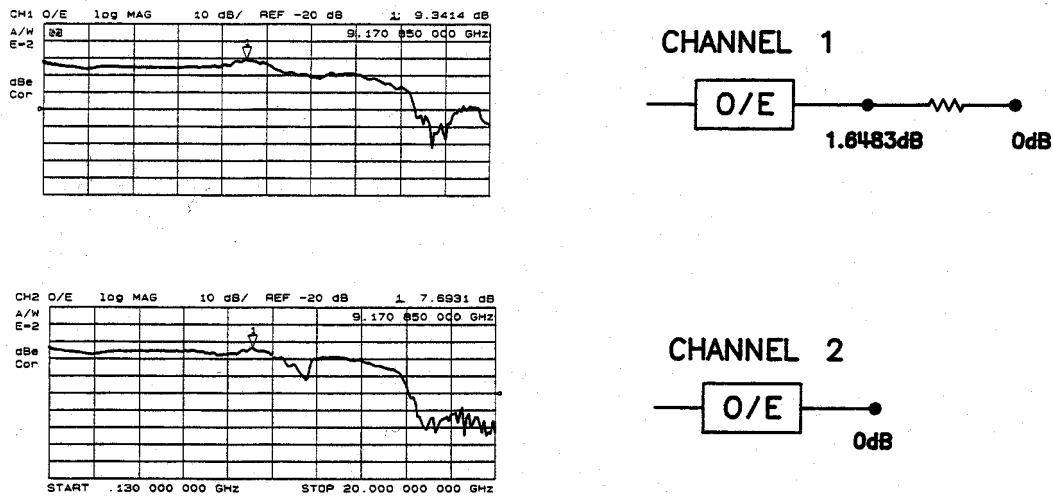
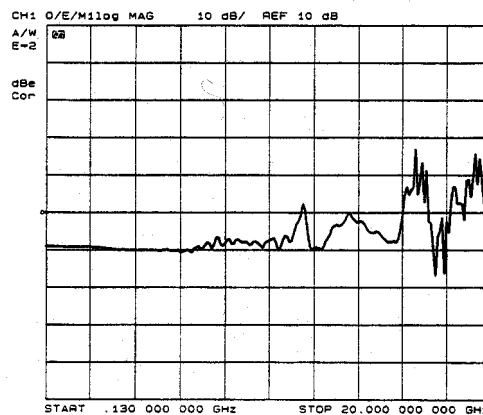


Figure 2-51. Ratio of the Receiver Measured on Channel 1 to the Receiver Measured on Channel 2



Modulation Phase

15. To return the analyzer display to the dual channel display and view the insertion phase response of both receivers, press:

Display, DUAL CHAN ON, DISPLAY: DATA

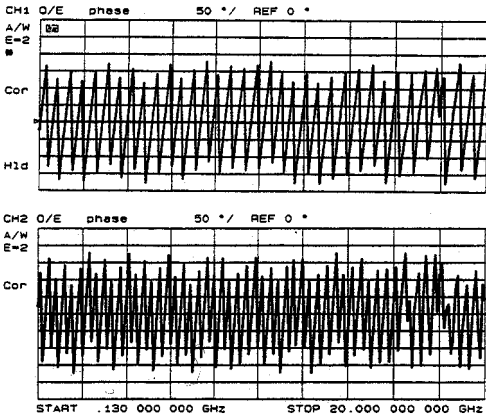
Format, PHASE

Chan 2, DISPLAY: MEMORY

Format, PHASE

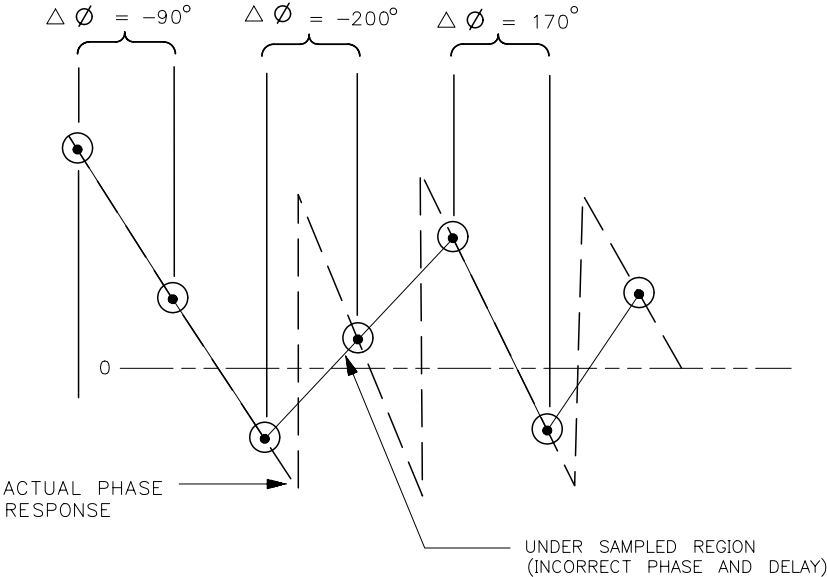
The figure below shows the modulation phase of the lightwave receivers. The analyzer measures and displays phase over the range of -180° to $+180^\circ$. As phase changes beyond these values, a sharp 360° transition occurs in the displayed data.

Figure 2-52. Modulation Phase Response of Receivers



The change in phase measured between two adjacent frequency points must be $<180^\circ$. If the phase is $>180^\circ$, incorrect delay information may result. The responses shown in Figure 2-52 have a change in phase $= >180^\circ$ between two adjacent frequency points. Figure 2-53 shows an example of phase samples being taken $<180^\circ$ apart.

Figure 2-53. Phase Samples $<180^\circ$ Apart



Electrical Length

The analyzer mathematically implements a function similar to the mechanical “line stretchers” of earlier analyzers. This feature simulates a variable length of lossless transmission line, which can be added to or removed from the analyzer’s internal reference port to compensate for the length of interconnecting cables. In this example, the electronic line stretcher is used to measure the electrical delay of the test devices.

The linearly changing phase, seen in [Figure 2-52](#), is due to the device’s electrical length. By adding electrical length (electrical delay) to the reference ports, the displayed phase shift can be reduced. Electrical length is then increased until the best flat line is achieved, and the device’s electrical length and electrical delay are read directly from the analyzer display. Since the phase samples taken are $>180^\circ$ apart with the current frequency span, a smaller frequency span is set up.

16. To narrow the span, press:

Center, 2, G/n

Span, 100 M/ μ

When measurement settings are changed, you must perform another calibration.

17. To perform a measurement calibration, connect thrus as shown in [Figure 2-48 on page 2-84](#), and press:

Cal, CALIBRATE MENU, RESPONSE, THRUS

When the calibration measurement is complete, the analyzer underlines the softkey label “THRUS”. Press **DONE:RESPONSE**.

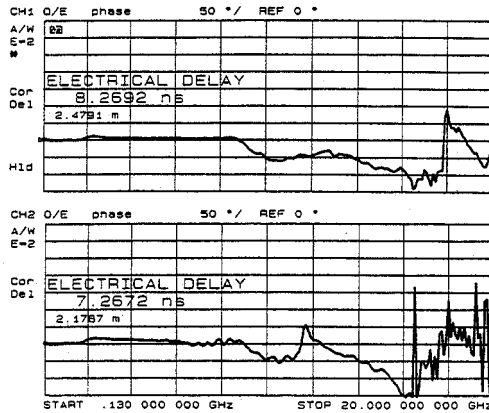
18. To save the measurement state and calibration in another register, press **Save/Recall, SAVE STATE**.

19. To activate the electrical delay function and add electrical length to each device, press **Scale Ref, ELECTRICAL DELAY**.

20. Turn the analyzer front panel knob to increase the electrical length until the best flat line is achieved. Refer to [“Electrical Length Added to Each Receiver” on page 2-89](#).

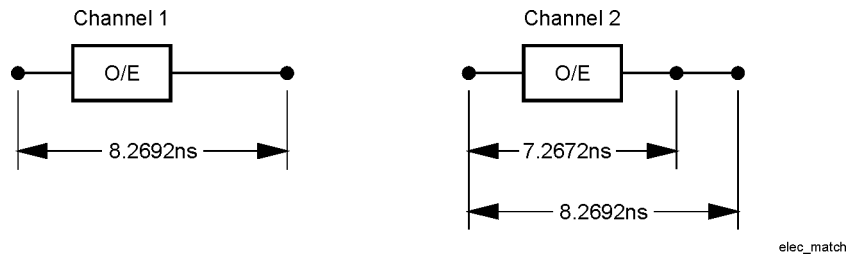
21. To activate channel 1 and increase the electrical length to the device measured on that channel, press **Chan 1, Scale Ref, ELECTRICAL DELAY** and turn the analyzer front panel knob to increase the electrical length until the best flat line is achieved.

Figure 2-54. Electrical Length Added to Each Receiver



Notice the difference between the electrical delay of the two receivers. If the devices were intended to be matched as closely as possible to each other, the device measured in channel 2 could have electrical length added to attain an electrical delay match. Refer to [Figure 2-55](#).

Figure 2-55. Electrical Length Added to Match Receivers



NOTE Although the receiver’s transfer characteristics can be matched with the addition of an attenuator to the device with higher power, the electrical delay will be altered with the addition of length. The electrical length must be measured again. It is possible that the addition of the attenuator to decrease the power may also be the length required to increase electrical delay.

22. Write down the delay of each device, this will be used in a later step.
23. To compare the receivers over their full range, press **Save/Recall, RECALL KEYS, RECALL KEYS MENU**, scroll to the register where you stored the measurement in Step 4, and press **RECALL REG**.
24. To add the electrical delay from step 22 to the device being measured on channel 2, press **Scale Ref, ELECTRICAL DELAY** and enter the delay.
25. To activate channel 1 and add the electrical delay from step 22 to the device being measured on channel 1, press **Chan 1, Scale Ref, ELECTRICAL DELAY** and enter the delay noted for the measurement in channel 1.

Use Trace Math to Compare the Electrical Length

26. To remove the extended length mathematically applied to the devices in the previous steps press

Scale Ref, ELECTRICAL DELAY, 0, x1

Chan 2, 0 x1

27. To input the current displayed data to memory 2 and activate channel 1, press:

Display, DATA->MEMORY2, Chan 1

28. To disengage the dual display to better view the receiver comparison, press **Display, DUAL/QUAD SETUP, DUAL CHAN OFF**.

29. To bring the trace stored on memory 2 (channel 2) to memory 1 (channel 1), press:

RETURN, SELECT DEFAULTS, MEMORY2->MEMORY1

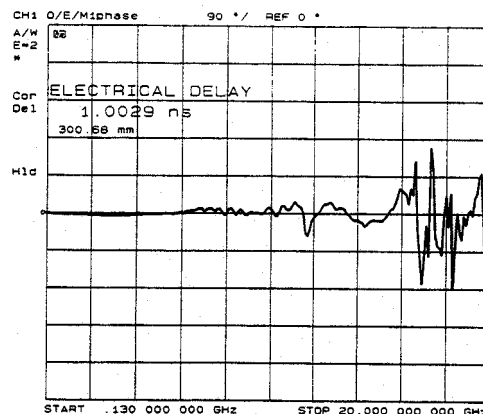
RETURN, MATH:, DATA/MEM

30. To add electrical delay to the result of the trace math in order to compare the electrical length of the devices, press **Scale Ref, ELECTRICAL DELAY**.

31. Turn the front panel knob to increase the electrical length until the best flat line is achieved. Refer to [“Receiver Electrical Length Comparison Using Trace Math”](#) on page 2-90.

The analyzer shows the ratio of the receiver response measured on channel 1 to the receiver response measured on channel 2.

Figure 2-56. Receiver Electrical Length Comparison Using Trace Math



Mathematically Combining Device Responses

The application described in this example is the separate response measurements of three devices (E/O, O/O, O/E) that are mathematically combined and then displayed. The three devices are connected together and the system response is measured for comparison.

By measuring each device of the system separately, you can determine the effect of individual devices.

CAUTION COSTLY REPLACEMENT of an entire lightwave assembly will result from damage to an optical port connector. Front panel test port connectors are integral parts of the lightwave components and ARE NOT separately replaceable. Keep optical cables connected to the test ports to protect the connectors from damage.

1. To set up the measurement, press:

Preset

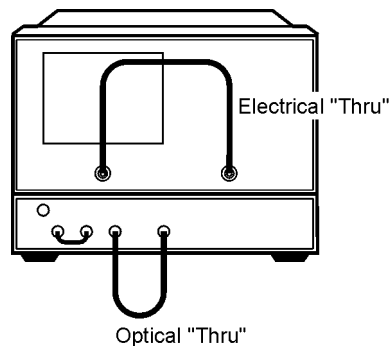
Meas, Trans:E/O

Avg, IF BW, 100 x1

(The bandwidth is decreased to filter out noise and lower the noise floor.)

2. Connect the electrical and optical thrus, as shown in [Figure 2-57](#).

Figure 2-57. Response Calibration Configuration for an E/O Transmission Measurement



oe_r_m3

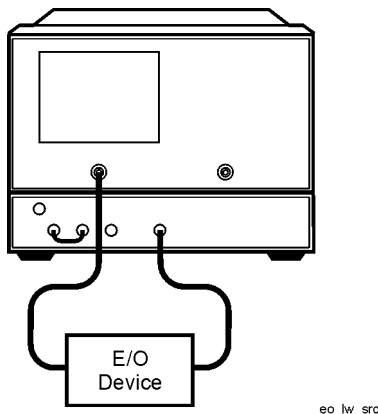
3. To perform a measurement calibration, press:

Cal, CALIBRATE MENU, RESPONSE, THRU, DONE:RESPONSE

When the calibration measurement is complete, the analyzer underlines the softkey label "THRU". Press **DONE**.

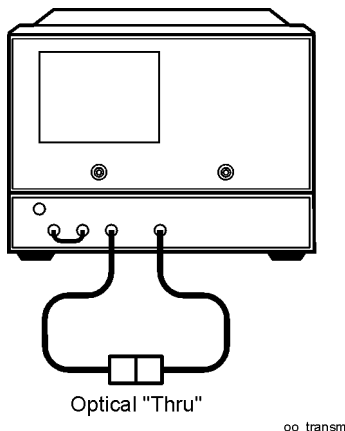
4. To save the measurement state and calibration in a register, press **Save/Recall, SAVE STATE**.
5. Insert the E/O device between the cables as shown in [Figure 2-58](#).

Figure 2-58. Lightwave Source Transmission Measurement Configuration



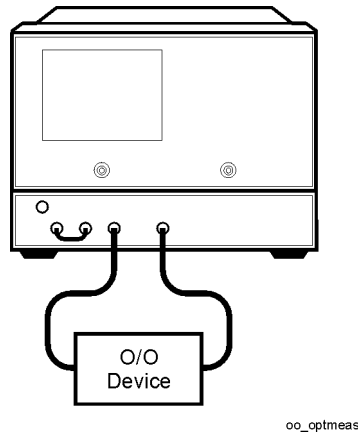
6. To more easily view the trace, press **Scale Ref, AUTO SCALE**.
7. To save the measurement trace in memory, press **Display, DATA->MEMORY 1**.
8. To set up the analyzer to measure the O/O component, press **Meas, Trans: O/O**.
9. Connect the optical thru between the optical test ports, as shown in [Figure 2-59](#).

Figure 2-59. Response Calibration Configuration for an O/O Transmission Measurement



10. To perform a response calibration, press:
Cal, CALIBRATE MENU, RESPONSE, THRU
When the calibration measurement is complete, the analyzer underlines the softkey label "THRU". Press **DONE: RESPONSE**.
11. To save the measurement state and calibration in a register, press **Save/Recall, SAVE STATE**.
12. Insert the optical device between the cables, as shown in [Figure 2-60](#).

Figure 2-60. Optical Transmission Measurement Configuration



13. To combine the measured trace with the memory trace, press:

Display, MATH:DATA/MEM, DATA*MEM

The displayed trace is the combination of the E/O and O/O device responses.

14. To save the displayed trace in memory, press:

RETURN, SELECT DEFAULTS, MATH->MEMORY 1

15. To set up the analyzer for an O/E measurement, press **Meas, Trans: O/E (Port 2)**.

16. Connect the optical thru between the optical test ports, as shown in [Figure 2-57](#).

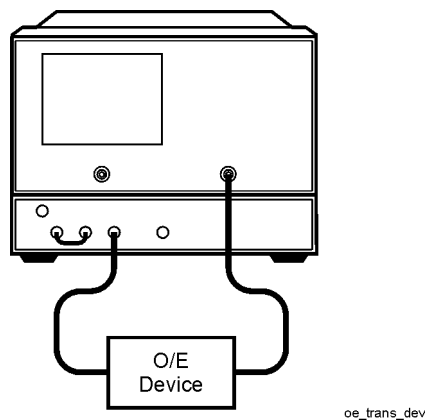
17. To perform a response calibration, press:

Cal, CALIBRATE MENU, RESPONSE, THRUS

18. When the calibration measurement is complete, the analyzer underlines the softkey label "THRUS". Press **DONE: RESPONSE**.

19. Insert the O/E device between the cables, as shown in [Figure 2-61](#).

Figure 2-61. O/E Transmission Measurement Configuration



20. To view the response of the O/E device, press **Display, DISPLAY: DATA**.

21. To combine the measurement trace with the already combined responses of the E/O and O/O devices, press **MATH: DATA*MEM**.

The displayed trace is the combination of all three device responses.

22. To set up the analyzer to measure the three devices together, press:

Meas, ELECTRICAL PARAMETERS, Trans: FWD S21 (B/R)

23. Connect the electrical cables between PORT 1 and PORT 2.

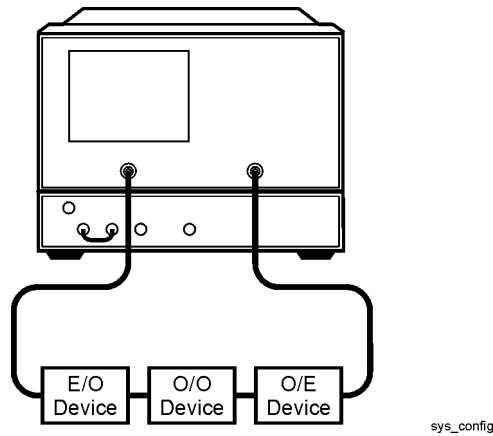
24. To perform a response calibration, press:

Cal, CALIBRATE MENU, RESPONSE, THRU

25. When the calibration measurement is complete, the analyzer underlines the softkey label "THRU". Press **DONE: RESPONSE**.

26. Insert all three devices between the electrical test ports, as shown in [Figure 2-62](#).

Figure 2-62. System Measurement Configuration



The trace displayed is the measurement of the combined device responses.

Using a Coefficient Model for Device Response and Design Model Comparison

The analyzer has coefficient model menus that make it possible to enter coefficients for a polynomial equation of the fourth order, describing device response versus frequency. The equation is saved in memory and is used as a model for some device or response function.

An application of the model is for simulation of a device that is not available. For example, you may want to measure an electro-optical link that includes an E/O and an O/E device. If you do not have one of the devices, the analyzer could simulate the device response by using a model. Then you can measure the device response and mathematically combine the results with the simulated response for a complete electro-optical link response. (See [“Mathematically Combining Device Responses”](#) on page 2-91 the previous measurement application for an example of mathematically combining responses.)

The application described in this example is the generation of a design model that is compared to a device measurement.

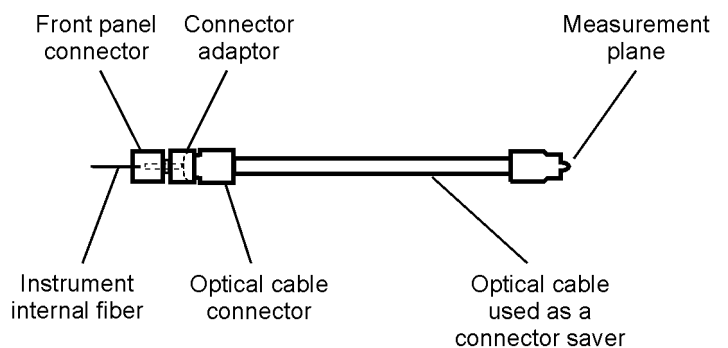
The polynomial equation has the form:

$$T(j\omega) = \frac{k \cdot e^{-j\omega Delay} (A_4(j\omega)^4 + A_3(j\omega)^3 + A_2(j\omega)^2 + A_1(j\omega) + 1)}{(B_4(j\omega)^4 + B_3(j\omega)^3 + B_2(j\omega)^2 + B_1(j\omega) + 1)}$$

where $j = \sqrt{-1}$

ω = frequency in radians

CAUTION COSTLY REPLACEMENT of an entire lightwave assembly will result from damage to an optical test port connector. Front panel test port connectors are integral parts of the lightwave components and ARE NOT separately replaceable. Keep optical cables connected to the test ports to protect the connectors from damage.



optcables

1. To set up the measurement, press:

Preset

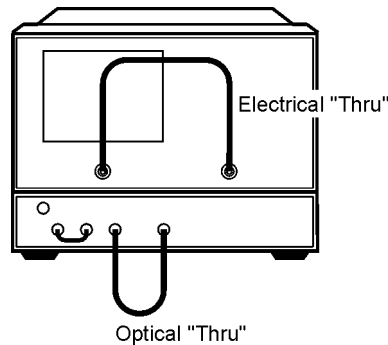
Meas, Trans: O/E

Avg, IF BW, 100, x1

(The IF bandwidth is decreased to filter out noise and lower the noise floor.)

2. Connect the electrical and optical thrus, as shown in [Figure 2-63](#).

Figure 2-63. Response Calibration Connection Configuration



oe_r_m3

3. To perform a measurement calibration, press:

Cal, CALIBRATE MENU, RESPONSE, THRU

When the calibration measurement is complete, the analyzer underlines the softkey label "THRU". Press **DONE: RESPONSE**.

To save the measurement state and calibration in a register, press **Save/Recall, SAVE STATE**.

4. To enter the coefficients for the design model, press:

Display, SELECT DEFAULTS, COEFFIC'NT MODEL MENU, NUMERATOR:k

5. Enter the multiplication constant in the numerator of the polynomial equation. Then press:

DELAY and enter the phase or delay factor. (The delay is a function of frequency.)

A1 and enter the first order coefficient in the numerator of the polynomial equation.

A2 and enter the second order coefficient in the numerator of the polynomial equation.

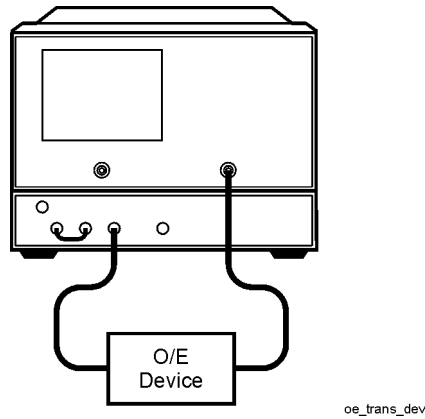
A3 and enter the third order coefficient in the numerator of the polynomial equation.

A4 and enter the fourth order coefficient in the numerator of the polynomial equation.

MORE to reach the second coefficient model menu.

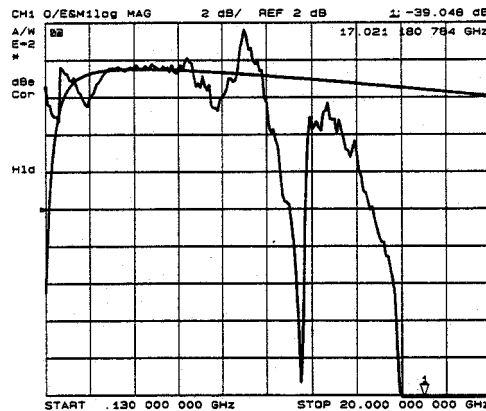
6. To enter the second half of the coefficient model, press:
DENOMIN: B1 and enter the first order coefficient in the denominator of the polynomial equation.
B2 and enter the second order coefficient in the denominator of the polynomial equation.
B3 and enter the third order coefficient in the denominator of the polynomial equation.
B4 and enter the fourth order coefficient in the denominator of the polynomial equation.
RETURN, RETURN, MODEL->MEMORY 1
7. To display the model and still be ready to view the measured device, press **Display**,
DISPLAY: DATA AND MEMORY.
8. Connect the device to be compared to the model, as shown in [Figure 2-64](#).

Figure 2-64. O/E Measurement Connection Configuration



[Figure 2-65](#) shows a typical display of a design model and a measured device.

Figure 2-65. Design Model and Measured Device



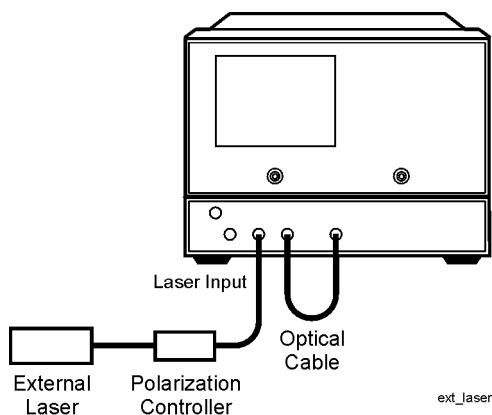
You can also set up limit testing to evaluate the measured device. See [“Using Limit Lines to Test a Device”](#) on page 2-42 for information on setting limit testing.

Using an External Laser

If you switch the measurement setup to include an external laser, follow this procedure to configure the system.

1. Press **Power**, **LASER INT EXT**.
2. Connect the external laser to the analyzer **EXTERNAL LASER INPUT** with an optical cable. Arrange the optical cable so it will not be moved during measurements.
3. Connect an optical thru between the analyzer **OPTICAL OUTPUT** and **OPTICAL RECEIVER** ports as shown in [Figure 2-66](#).

Figure 2-66. Configuration for Adjusting an External Laser



4. Press **Sweep Setup**, **CW FREQ**, **1, G/n**.
5. Adjust the external laser's polarization controller to achieve the maximum signal level. (If the polarization controller is a multiple element type, all elements must be adjusted.)
6. Press **System**, **SERVICE MENU**, **TESTS**, **LIGHTWAVE TESTS**, **EXECUTE TEST**.
7. Repeat steps 4, 5, and 6 until the signal level reaches a uniformly high level.

3

[“Optimizing Measurement Results” on page 3-2](#)

[“Increasing Measurement Accuracy” on page 3-4](#)

[“Increasing Dynamic Range” on page 3-7](#)

[“Reducing Noise” on page 3-8](#)

[“Reducing Receiver Crosstalk” on page 3-9](#)

Optimizing Measurement Results

Optimizing Measurement Results

This chapter offers various methods that you can use to get the necessary results for your measurement application. Because the analyzer settings can be interactive, you need to consider a balance between the accuracy and speed of a measurement.

Taking Care of Connectors

Having good connector care and connection techniques extends the life of these devices. In addition, you obtain the most accurate measurements.

For information about connector care, see Chapter 1.

Prior to making connections to the lightwave component analyzer, carefully review the information about inspecting and cleaning connectors.

For additional connector care instruction, contact your local Agilent Technologies Sales and Service Office about course numbers HP/Agilent 85050A+24A and 85050A+24D.

Increasing Measurement Accuracy

The following all situations contribute to loss of accuracy in a measurement.

Interconnecting Cables

Cables that connect the device under test (DUT) to the analyzer are often the most significant contribution to random errors of your measurement. You should frequently perform the following steps as a precaution against errors caused by cable interconnections:

- Inspect for lossy cables.
- Inspect for damaged cable connectors.
- Practice good connector care techniques.
- Minimize cable position changes between error-correction and measurements.
- Inspect for cables which dramatically change magnitude or phase response when flexed. (This may indicate an intermittent problem.)

Improper Calibration Techniques

Calibrations techniques performed improperly contribute to random errors to your measurement. You should frequently perform the following steps as a precaution against errors caused by calibration techniques:

- Verify the correct calibration kit definition is selected.
- Verify the correct standards have been connected.
- Verify the good condition of the calibration standards.

Connector Repeatability

Connector repeatability is a source of random measurement error. Measurement error-corrections do not compensate for these errors. For all connectors, you should frequently perform the following steps as a precaution against errors caused by connector repeatability:

- Inspect the connectors.
- Clean the connectors.
- Gauge the electrical connectors.
- Use correct connection techniques. (Refer to [Chapter 1, “Caring for Electrical Connections”](#).)

Temperature Drift

Electrical characteristics will change with temperature due to the thermal expansion characteristics of devices within the analyzer, calibration devices, test devices, cables, and

adapters. Therefore, the operating temperature is a critical factor in their performance. During a measurement calibration, the temperature of the calibration devices must be stable and within 25 ± 5 °C.

- Use a temperature-controlled environment.
- Ensure the temperature stability of the calibration devices.
- Avoid handling the calibration devices unnecessarily during calibration.
- Ensure the ambient temperature is ± 1 °C of measurement error-correction temperature.

Frequency Drift

Minute changes in frequency accuracy and stability can occur as a result of temperature and aging (on the order of parts per million). If you require greater frequency accuracy, override the internal crystal with a high-stability external source, frequency standard.

Operation Verification

You should periodically check the accuracy of the analyzer measurements, by measuring the verification device in the N1011A Verification Kit. You should also contact Agilent to verify all of the instrument's published specifications once per year.

Reference Plane and Port Extensions

Use the port extension feature to compensate for the phase shift of an extended measurement reference plane, due to such additions as cables, adapters, and fixtures, after completing an error-correction procedure (or when there is no active correction).

Using port extensions is similar to using electrical delay. However, using port extensions is the preferred method of compensating for test fixture phase shift. The following table explains the difference between port extensions and electrical delay.

Table 3-1. Differences between PORT EXTENSIONS and ELECTRICAL DELAY

	PORT EXTENSIONS	ELECTRICAL DELAY
Main Effect	The end of a cable becomes the test port plane for all measurements.	Compensates for the electrical length of a cable. Set the cable's electrical length $\times 1$ for transmission. Set the cable's electrical length $\times 2$ for reflection.
Measurements Affected	All measurements.	Only the currently selected measurements.
Electrical Compensation	Intelligently compensates for 1 times or 2 times the cable's electrical delay, depending on which measurement is computed.	Only compensates for electrical length.

You can activate a port extension by pressing **Cal, MORE, PORT EXTENSIONS, EXTENSIONS ON**. Then enter the delay to the reference plane.

Increasing Dynamic Range

Dynamic range is the difference between the analyzer's maximum allowable input level and minimum measurable power. For a measurement to be valid, input signals must be within these boundaries. The dynamic range is affected by these factors:

- test port input power
- test port noise floor
- receiver crosstalk

Increase the Test Port Input Power

You can increase the analyzer's source output power so that the test device output power is at the top of the measurement range of the analyzer test port.

Press **Power**, and enter the new source power level, followed by **x1**.

CAUTION Do not exceed the maximum test port power level that is printed on the front panel of your analyzer. Exceeding this maximum power level may damage your analyzer.

Reduce the Receiver Noise Floor

Refer to [“Reducing Noise” on page 3-8](#).

Reduce the Receiver Crosstalk

Refer to [“Reducing Receiver Crosstalk” on page 3-9](#).

Reducing Noise

You can use two analyzer functions to help reduce the effect of noise on the data trace:

- activate measurement averaging
- reduce system bandwidth

To Activate Averaging

The noise is reduced with each new sweep as the effective averaging factor increments.

1. Press **Avg**, **AVERAGING FACTOR**.
2. Enter a value followed by **x1**.
3. Press **AVERAGING ON**.

Refer to [“Averaging” on page 5-28](#) in the Reference for more information.

To Change System Bandwidth

By reducing the system bandwidth, you reduce the noise that is measured during the sweep. While averaging requires multiple sweeps to reduce noise, narrowing the system bandwidth reduces the noise on each sweep, however, the sweep will be slower.

1. Press **Avg**, **IF BW**.
2. Enter the IF bandwidth value that you want, followed by **x1**.

Narrower system bandwidths cause longer sweep times. When in auto sweep time mode, the analyzer uses the fastest sweep time possible for any selected system bandwidth. Auto sweep time mode is the default (preset) analyzer setting.

NOTE Another capability that can be used for effective noise reduction is the marker statistics function, which computes the average value of part or all of the formatted trace.

Reducing Receiver Crosstalk

To reduce receiver crosstalk you can do the following:

- Perform a response and isolation measurement calibration.
- Set the sweep to the alternate mode.

Alternate sweep is intended for measuring wide dynamic range devices, such as high pass and bandpass filters. This sweep mode removes a type of leakage term through the device under test, from one channel to another.

To set the alternate sweep, press **Cal, MORE, ALTERNATE A and B.**

Refer to [“Procedures for Error Correcting Your Measurements”](#) on page 4-9.

4

- “Calibrating for Increased Measurement Accuracy” on page 4-2
- “Calibration Considerations” on page 4-3
- “Procedures for Error Correcting Your Measurements” on page 4-9
- “O/O Response Calibration” on page 4-11
- “O/O Response and Isolation Calibration” on page 4-13
- “E/O Response Calibration” on page 4-16
- “E/O Reflection Sensitivity Calibration” on page 4-17
- “E/O Response and Isolation Calibration” on page 4-18
- “E/O Response and Match Calibration” on page 4-20
- “O/E Response Calibration” on page 4-23
- “O/E Response and Isolation Calibration” on page 4-24
- “O/E Response and Match Calibration” on page 4-26
- “Modifying Lightwave Calibration Kits” on page 4-29
- “Modifying an Electrical Calibration Kit” on page 4-30
- “Verify Performance” on page 4-39

Calibrating for Increased Measurement Accuracy

Calibrating for Increased Measurement Accuracy

Measurement accuracy is greatly influenced by factors external to the lightwave component analyzer. Components of the measurement setup, such as interconnecting cables and adapters, introduce variations in magnitude and phase that can mask the actual response of the device under test.

This chapter contains various measurement calibration procedures that you can use to increase measurement accuracy by removing systematic errors (repeatable measurement variations) in the test setup. During these procedures, the analyzer measures known standard devices, and uses the results of these measurements to characterize the system.

Measurement accuracy and system characteristics can be affected by the following factors:

- Adapting to a different connector type or impedance.
- Connecting a cable between the test device and an analyzer test port.
- Connecting any attenuator or other such device on the input or output of the test device.

If your test setup meets any of the these conditions, the following system characteristics may be affected:

- amplitude at device input
- frequency response accuracy
- directivity
- crosstalk (isolation)
- source match
- load match

Calibration Considerations

Measurement Parameters

Calibration procedures are parameter-specific, rather than channel-specific. When a parameter is selected, the instrument checks the available calibration data, and uses the data found for that parameter. For example, if a transmission response calibration is performed for B/R, and an S_{11} 1-port calibration for A/R, the analyzer retains both calibration sets and corrects whichever parameter is displayed. Once a calibration has been performed for a specific parameter or input, measurements of that parameter remain calibrated in either channel, as long as stimulus values are coupled. In the response and response and isolation calibrations, the parameter must be selected before calibration. Other correction procedures select parameters automatically. Changing channels during a calibration procedure invalidates the part of the procedure already performed.

Device Measurements

In calibration procedures that require measurement of several different devices, for example a short, an open, and a load, the order in which the devices are measured is not critical. Any standard can be re-measured, until the **DONE**, key is pressed. The change in trace during measurement of a standard is normal.

Clarifying Type-N Connector Sex

When you are performing error correction for a system that has type-N port connectors, the softkey menus label the sex of the test port connector – *not* the calibration standard connector. For example, the label, SHORT (F), refers to the short that will be connected to the female test port. Since many devices have type-N (f) connectors, the calibration standard to select is the type-N (m). Be sure to use the port extension pin on the type-N (m) Open calibration standard.

NOTE Since the 85032F calibration kit offsets are equal for both male and female connectors, the standard's sex type is not requested during the calibration.

Omitting Isolation Calibration

Isolation calibration can be omitted for most measurements, except where high dynamic range is a consideration. Use the following guidelines.

- When the required dynamic range is less than 90 dB: Omit isolation calibration for most measurements.
- When the required dynamic range is 90 to 100 dB: Isolation calibration is recommended with test port power greater than 0 dBm. For this isolation calibration, averaging should be turned on with an averaging factor at least four times the measurement averaging factor. For example, use an averaging factor of 16 for the isolation calibration, and then reduce the averaging factor to four for the measurement after calibration.

- When the required dynamic range is greater than 100 dB: Same as 90 to 100 dB, but alternate mode should be used. See [“Increasing Dynamic Range” on page 3-7](#).

Saving Calibration Data

You should save the calibration data, either in the internal non-volatile memory or on a disk. If you do not save it, it will be lost if you select another calibration procedure for the same channel, or if you change stimulus values. Instrument preset, power on, and instrument state recall will also clear the calibration data.

Restarting a Calibration

If you interrupt a calibration to go to another menu, such as averaging, you can continue the calibration by pressing the **RESUME CAL SEQUENCE**, softkey in the correction menu.

The Calibration Standards

During measurement calibration, the analyzer measures actual, well-defined standards and mathematically compares the results with ideal "models" of those standards. The differences are separated into error terms which are later removed during error correction. Most of the differences are due to systematic errors—repeatable errors introduced by the analyzer, test set, and cables—which are correctable.

The standard devices required for a measurement calibration are optical reflectors, Fresnel reflectors, optical thru, and electrical standards available in compatible calibration kits with different connector types. Electrical calibration kits contain at least one short circuit, one open circuit, and an impedance-matched load. In kits that require adapters for interface to the test set ports, the adapters are phase-matched for calibration prior to measurement of non-insertable and non-reversible devices. Other standard devices can be used by specifying their characteristics in a user-defined kit, as described in [“Modifying Lightwave Calibration Kits” on page 4-29](#) and in [“Modifying an Electrical Calibration Kit” on page 4-30](#).

The accuracy improvement of the correction is limited by the quality of the standard devices, and by the connection techniques used. For maximum accuracy, ensure that the connectors are clean and connections are tight.

Frequency Response of Calibration Standards

In order for the response of a reference standard to show as a dot on the smith chart display format, it must have no phase shift with respect to frequency. Standards that exhibit such "perfect" response are the following:

- 7-mm short (with no offset)
- type-N male short (with no offset)

There are two reasons why other types of reference standards show phase shift after calibration:

- The reference plane of the standard is electrically offset from the mating plane of the test port. Such devices exhibit the properties of a small length of transmission line, including a certain amount of phase shift.

- The standard is an open termination, which by definition exhibits a certain amount of fringe capacitance (and therefore phase shift). Open terminations which are offset from the mating plane will exhibit a phase shift due to the offset in addition to the phase shift caused by the fringe capacitance.

The most important point to remember is that these properties will not affect your measurements. The analyzer compensates for them during measurement. As a result, if these standards are measured after a calibration, they will not appear to be "perfect" shorts or opens. This is an indication that *your analyzer is working properly* and that it has successfully performed a calibration. [Figure 4-1](#) shows sample displays of various calibration standards after calibration.

Electrical Offset

Some standards have reference planes that are electrically offset from the mating plane of the test port. These devices will show a phase shift with respect to frequency. [Figure 4-1](#) shows which reference devices exhibit an electrical offset phase shift. The amount of phase shift can be calculated with the formula:

$$\Phi = (360^\circ \times f \times l) / c \text{ where:}$$

f = frequency

l = electrical length of the offset

c = speed of light (3×10^8 meters/second)

Fringe Capacitance

All open circuit terminations exhibit a phase shift over frequency due to fringe capacitance. Offset open circuits have increased phase shift because the offset acts as a small length of transmission line. Refer to [Table 4-1](#).

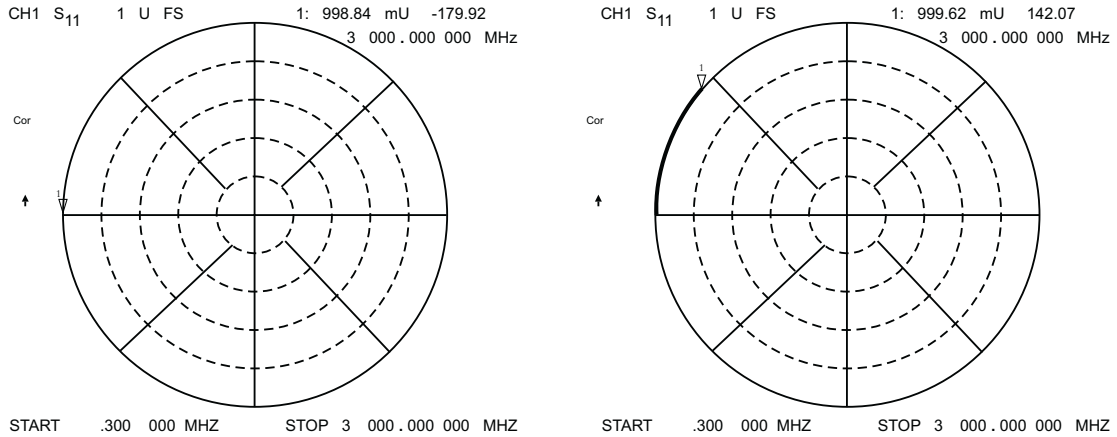
Table 4-1. Calibration Standard Types and Expected Phase Shift

Test Port Connector Type	Standard Type	Expected Phase Shift
7-mm Type-N male	Short	180°
3.5-mm male 3.5-mm female 2.4-mm male 2.4-mm female Type-N female 75Ω Type-N female	Offset Short	$180^\circ + \frac{(360^\circ \times f \times l)}{c}$
7-mm Type N-male	Open	$0^\circ + \Phi^\circ_{\text{capitance}}$

Table 4-1. Calibration Standard Types and Expected Phase Shift

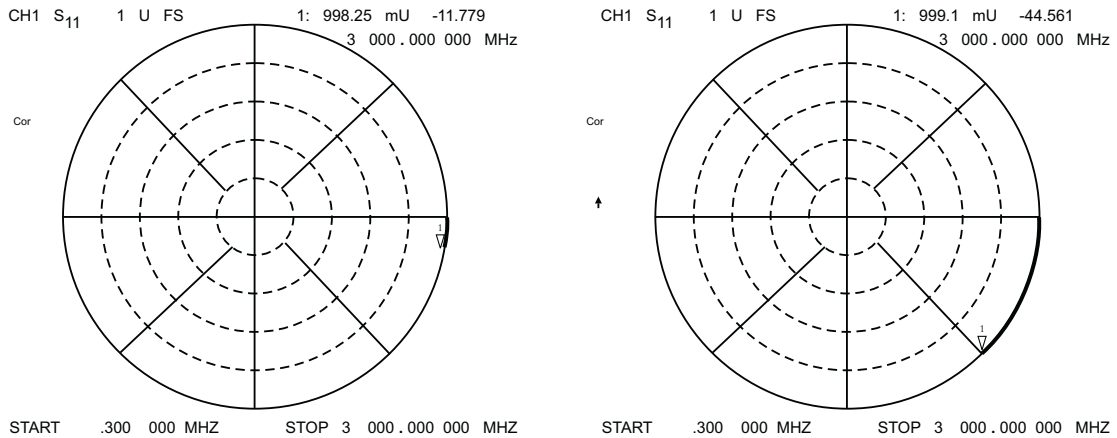
Test Port Connector Type	Standard Type	Expected Phase Shift
3.5-mm male 3.5-mm female 2.4-mm male 2.4-mm female Type-N female 75Ω Type-N female	Offset Open	$0^\circ + \Phi^\circ_{\text{capitance}} + \frac{(360^\circ \times f \times l)}{c}$

Figure 4-1. Typical Responses of Calibration Standards after Calibration



**7mm or Type-N Male
 Short (No Offset)**

**Type-N Female,
 3.5mm Male or Female Offset Short**



**7mm or Type-N Male
 Open (No Offset) with Fringing Capacitance**

**Type-N Female
 3.5 mm Male or Female Offset Open**

pa5162e

Interpolated Error Correction

You may want to use interpolated error correction when you choose a subset of a frequency range that you already corrected, when you change the number of points, or when you change to CW. This feature also allows you to change the parameters in a 2-port correction, such as IF bandwidth, power, or sweep time. The analyzer calculates the systematic errors from the errors of the original correction.

To activate interpolated measurement correction, press **Cal, INTERPOL ON off**, so that ON is selected, and **CORRECTION on OFF**, so that ON is selected. When interpolation is in use, the notation $C\Delta$ will appear on the analyzer display.

NOTE The preset state of the instrument can be configured so that interpolated error correction is on or off. Press **System, CONFIGURE MENU, USER SETTINGS, PRESET SETTINGS, CAL INTERP ON off**, to configure the preset state of interpolated error correction.

System performance is unspecified when using interpolated error correction. The quality of the interpolated error correction is dependent on the amount of phase shift and the amplitude change between measurement points. If phase shift is no greater than 180° per approximately five measurement points, interpolated error correction offers a great improvement over uncorrected measurements. The accuracy of interpolated error correction improves as the phase shift and amplitude change between adjacent points decrease. When you use the analyzer in linear frequency sweep, perform the original calibration with at least 30 points per 1 GHz of frequency span for greatest accuracy with interpolated error correction.

Interpolated error correction is available in three sweep modes: linear frequency, power sweep, and CW time.

NOTE If there is a valid correction array for a linear frequency sweep, this may be interpolated to provide correction at the CW frequency used in power sweep or CW time modes. This correction is part of the interpolated error-correction feature.

Error-Correction Stimulus State

Error correction is only valid for a specific stimulus state, which you must select before you start a correction. If you change any of the following parameters, you will invalidate the correction and the analyzer will switch the correction off (unless the interpolated error correction feature is activated):

- frequency range
- number of points
- sweep type

The error correction quality may be degraded (Cor changes to CΔ), if you change the following stimulus state parameters:

- sweep time
- system bandwidth
- output power

If correction is turned off or shows CΔ, the original stimulus state can be recalled by, first, turning interpolation off (**INTERPOL ON off**), and toggling correction off and then on (**CORRECTION ON off**).

Procedures for Error Correcting Your Measurements

This section has example procedures or information on the following topics:

- frequency response correction
- frequency response and isolation correction
- frequency response and match (E/O and O/E devices)

NOTE If the channels are uncoupled, you must make a correction for each channel.

Types of Error Correction

Several types of error correction are available that remove from one to twelve systematic errors. [Table 4-1](#) explains each correction and its uses.

Table 4-1. Purpose and Use of Different Calibration Procedures

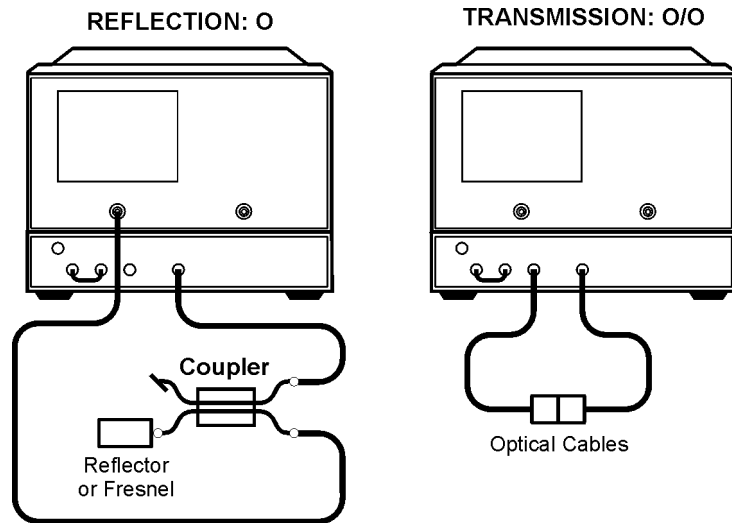
Calibration Procedure	Corresponding Measurement	Errors Removed	Standard Procedure
Optical (O, O/O)			
Response	Transmission or reflection measurement when the highest accuracy is not required.	Frequency response	Optical thru for transmission, Fresnel or Reflector for reflection
Response & Isolation	Transmission of high insertion loss devices or reflection of high return loss devices.	Frequency response, plus isolation in transmission or directivity in reflection.	Same as response plus disconnect cable or turn off laser.
Optical (O/E)			
Response	Transmission measurement	Frequency response	Optical and/or electrical thrus
Response & Isolation	Transmission of high Insertion loss devices	Frequency response plus isolation	Same as Response plus disconnect cable or turn off laser.
Response & Match	Transmission measurement for devices with large electrical reflectivity.	Frequency response plus electrical mismatch	Same as Response plus short, opens, and loads.
Optical (E/O)			
Response	Transmission or reflection sensitivity measurement.	Frequency response	Optical and /or electrical thrus for transmission, optical load for reflection sensitivity
Response and Isolation	Transmission of high insertion loss devices.	Frequency response plus isolation	Same as response plus disconnect the cable or turn off the laser.
Response & Match	Transmission measurement for devices with large electrical reflectivity.	Frequency response plus electrical mismatch	Same as Response plus short, opens, and loads.
Note: If cables, connectors, or adapters are removed from the measurement setup that were used in the calibration, their effect must be accounted for by adding a port extension equivalent to the electrical length of the missing component(s).			

O/O Response Calibration

By following this procedure, you can effectively remove the frequency response errors of the test setup for reflection or transmission measurements of optical devices.

1. Configure the equipment to include any cables that will be used in the device measurement.
2. Press **Meas**, and choose the type of measurement:
 - If you want to measure the reflection of an optical device, press **Refl: O**.
 - If you want to measure the transmission of an optical device, press **Trans: O/O**.
3. Select any other settings that you want for the device measurement: power, sweep type, number of points, or IF bandwidth.
4. Press **Cal**, **CALIBRATE MENU**, **RESPONSE**.
5. Connect the calibration standard as shown:
 - If you are making a reflection measurement, either connect a 100% reflector or leave unterminated for a Fresnel reflection.
 - If you are making a transmission measurement, connect optical cables as shown.

Figure 4-2. O/O Response Calibration Setup



oo_rescal

6. Measure the calibration standard:

- If you are making a reflection measurement, press **REFLECTOR** or **FRESNEL**, depending on the standard you are using.
 - If you are making a transmission measurement, press **THRU**.
7. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The analyzer underlines the softkey that you selected after it finishes the measurement, and computes the error coefficients. Press **DONE: RESPONSE**.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to “Saving and Recalling Measurements and Data.”

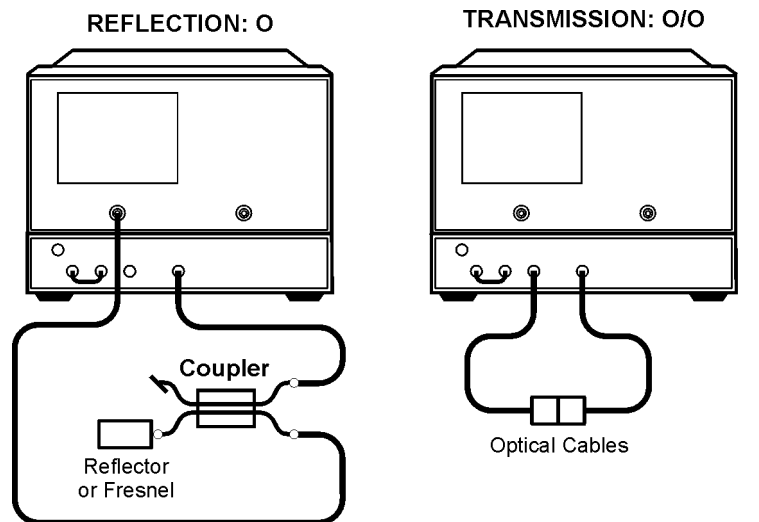
This completes the response correction for optical measurements. You can connect and measure your device under test.

O/O Response and Isolation Calibration

By following this procedure, you can effectively remove the frequency response and isolation errors for transmission measurements or the directivity error in reflection measurements of devices with wide dynamic range.

1. Configure the equipment to include any cables that will be used in the device measurement.
2. Press **Meas**, and choose the type of measurement:
 - If you want to measure the reflection of an optical device, press **Refl: O**.
 - If you want to measure the transmission of an optical device, press **Trans: O/O**.
3. Press **Avg, IF BW, 100, x1** to reduce the the IF bandwidth (10 Hz IF BW for maximum dynamic range).
4. Select any other settings that you want for the device measurement: power, sweep type, or number of points.
5. Press **Cal, CALIBRATE MENU, RESPONSE & ISOL'N, RESPONSE**.
6. Connect the standard as shown:
 - If you are making a reflection measurement, either connect a 100% reflector or leave unterminated for a Fresnel reflection.
 - If you are making a transmission measurement, connect an optical cables as shown.

Figure 4-3. O/O Response Calibration Setup



oo_rescal

7. Measure the calibration standard:

- If you are making a reflection measurement, press **REFLECTOR** or **FRESNEL**, depending on the standard you are using.
- If you are making a transmission measurement, press **THRU**.

8. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The analyzer underlines the softkey that you selected after it finishes the measurement.

9. Press **DONE: RESPONSE**.

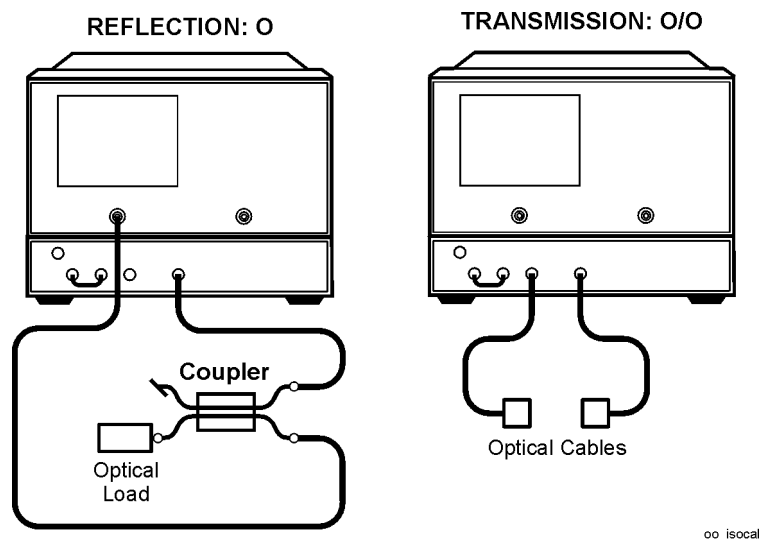
10. Press **Avg, AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)

11. Press **Cal, RESUME CAL SEQUENCE**.

12. Connect the isolation standard as shown:

- If you are making a reflection measurement, connect the optical load.
- If you are making a transmission measurement, switch off the laser or break the connection between the optical output and optical input.

Figure 4-4. O/O Isolation Calibration Setup



oo_isocal

13. Press **ISOL'N STD**. The optical isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.
14. Press **Avg, AVERAGING OFF, Cal, RESUME CAL SEQUENCE, DONE: RESP ISOL'N CAL**. The frequency response and isolation error coefficients are computed and stored. A corrected trace is displayed and the notation "Cor" appears at the left side of the screen.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

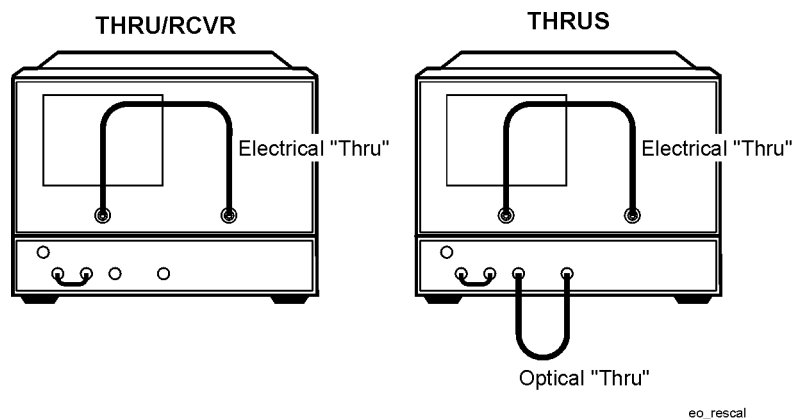
This completes the response and isolation correction for optical measurements. You can connect and measure your device under test.

E/O Response Calibration

By following this procedure, you can effectively remove the frequency response errors of the test setup for transmission measurements of electrical-to-optical devices.

1. Press **Meas, Trans: E/O**.
2. Select any other measurement settings that you want for the device measurement: power, sweep type, number of points, or IF bandwidth.
3. Press **Cal, CALIBRATE MENU, RESPONSE**.
4. Connect the calibration standards as shown in [Figure 4-5](#). If you are using the analyzer internal laser as a source for modulating the device under test, use the THRU/RCVR configuration. Otherwise, use the THRU configuration.

Figure 4-5. E/O Response Calibration Setup



5. Measure the calibration standards:
 - If you connected both the electrical and optical thrus, press **THRUS**.
 - If you only connected the electrical thru, press **THRU/RCVR**.
6. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The analyzer underlines the softkey that you selected after it finishes the measurement, and computes the error coefficients. Press **DONE: RESPONSE**.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

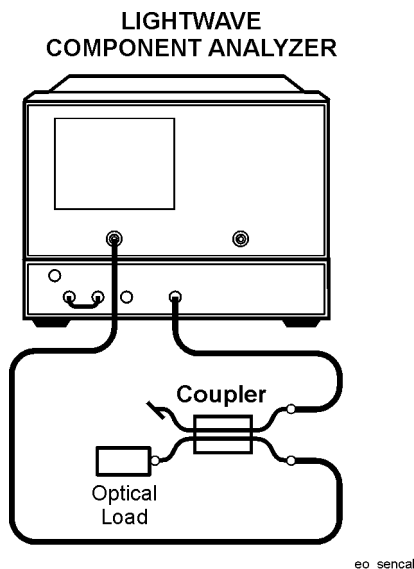
This completes the response correction for electrical-to-optical device measurements. You can connect and measure your device under test.

E/O Reflection Sensitivity Calibration

By following this procedure, you can effectively remove the frequency response errors of the test setup for a reflection sensitivity measurement of electrical-to-optical devices. This calibration helps you determine the sensitivity of a modulated laser to its termination. Using the external 10 dB coupler, approximately 90% of the power returns to the device under test.

1. Press **Meas, Trans: E/O**.
2. Select any settings that you want for the device measurement: power, sweep type, number of points, or IF bandwidth.
3. Press **Cal, CALIBRATE MENU, RESPONSE**.
4. Connect the calibration standards as shown:

Figure 4-6. E/O Reflection Sensitivity Calibration Setup



5. Press **REFL SENS**.
6. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The analyzer underlines the softkey that you selected after it finishes the measurement, and computes the error coefficients. Press **DONE: RESPONSE**.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

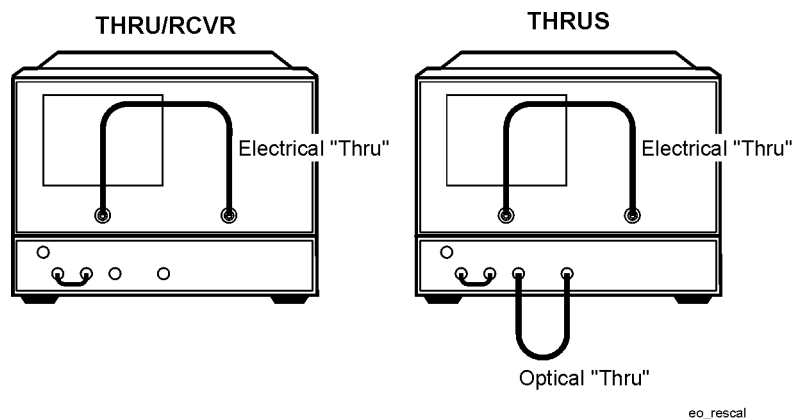
This completes the reflection sensitivity calibration for electrical-to-optical device measurements. You can connect and measure your device under test

E/O Response and Isolation Calibration

By following this procedure, you can effectively remove the frequency response and isolation errors for transmission measurements or the directivity error in reflection measurements of electrical-to-optical devices with wide dynamic range.

1. Press **Meas, Trans: E/O**.
2. Press **Avg, IF BW, 100, x1** to reduce the the IF bandwidth (10 Hz IF BW for maximum dynamic range).
3. Select any settings that you want for the device measurement: power, sweep type, or number of points.
4. Press **Cal, CALIBRATE MENU, RESPONSE & ISOL'N, RESPONSE**.
5. Connect the calibration standards as shown in [Figure 4-7](#). If you are using the analyzer internal laser as a source for modulating the device under test, use the THRU/RCVR configuration. Otherwise, use the THRUS configuration.

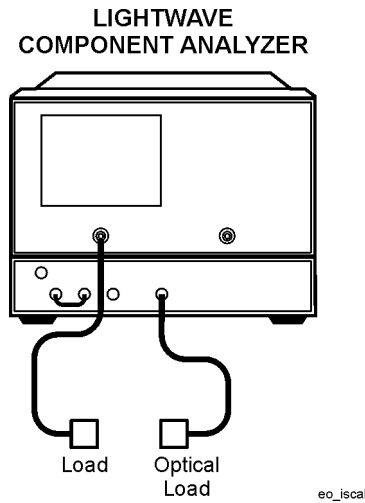
Figure 4-7. E/O Response Calibration Setup



6. Measure the calibration standards:
 - If you connected both the electrical and optical thrus, press **THRUS**.
 - If you only connected the electrical thru, press **THRU/RCVR**.
7. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. Press **DONE: RESPONSE**.
8. Press **Avg, AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)
9. Press **Cal, RESUME CAL SEQUENCE**.

10. Connect the isolation standards as shown in [Figure 4-8](#).

Figure 4-8. E/O Isolation Calibration Setup



11. Press **ISOL'N STD**. The E/O isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.

12. Press **Avg**, **AVERAGING OFF**, **Cal**, **RESUME CAL SEQUENCE**, **DONE: RESP ISOL'N CAL**. The frequency response and isolation error coefficients are computed and stored. A corrected trace is displayed and the notation "Cor" appears at the left side of the screen.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

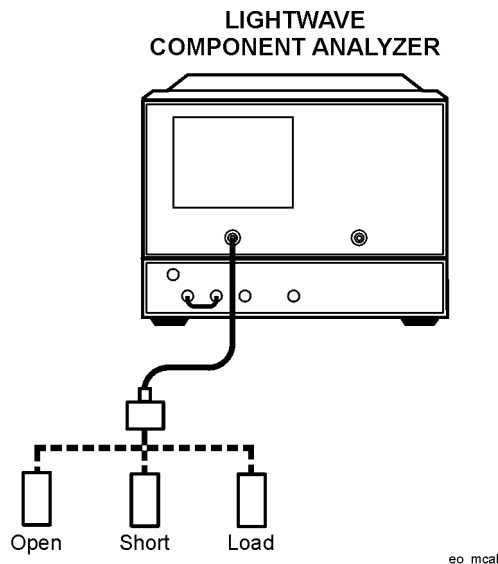
This completes the response and isolation correction for electrical-to-optical device measurements. You can connect and measure your device under test.

E/O Response and Match Calibration

By following this procedure, you can effectively remove the frequency response and port match errors of the test setup for transmission measurements of modulated lasers.

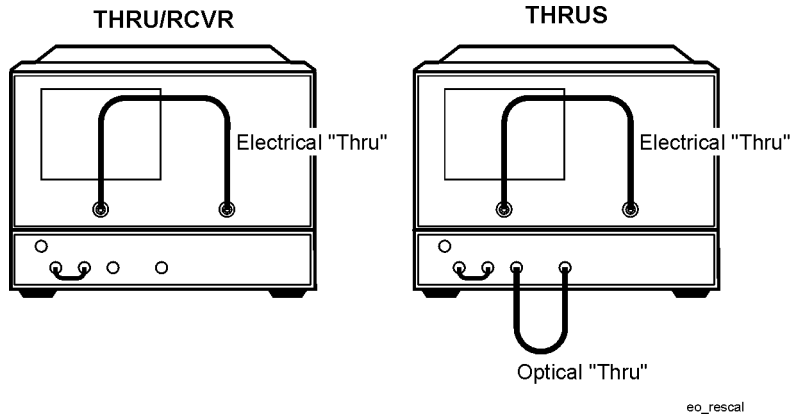
1. Press **Meas, Trans: E/O**.
2. Press **Avg, IF BW, 100, x1** to reduce the the IF bandwidth (10 Hz IF BW for maximum dynamic range).
3. Select any other settings that you want for the device measurement: power, sweep type, or number of points.
4. Press **Cal, CALIBRATE MENU, RESPONSE & MATCH, REFLECTION**.
5. Connect the open calibration standard to electrical PORT 1 as shown in [Figure 4-9](#).

Figure 4-9. Reflection Measurements for E/O Response and Match Calibration



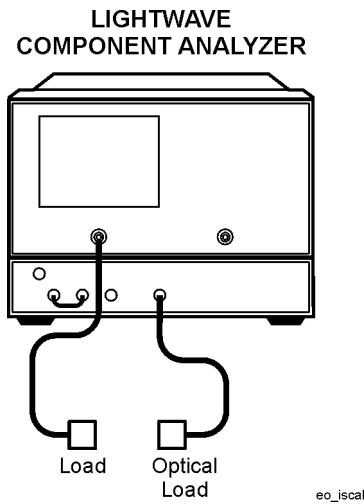
6. Measure the calibration standard by pressing **FORWARD: OPEN**.
7. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The softkey label is then underlined.
8. Repeat steps 5 through 7 to measure the short and the load calibration standards.
9. Press **DONE: LOADS, STANDARDS DONE**.
10. Connect the cables, as shown in [Figure 4-10](#). If you are using the analyzer internal laser as a source for modulating the device under test, use the THRU/RCVR configuration. Otherwise, use the THRU configuration.

Figure 4-10. Transmission Measurements for E/O Response and Match Calibration



11. To measure the calibration standards, press **FWD TRANS THRUS+RCVR**.
 - If you connected both the electrical and optical thru, press **THRU**.
 - If you only connected the electrical thru, press **THRU/RCVR**.
12. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The softkey label is then underlined. Press **DONE: THRUS+RCVR**.
13. If you do not want the isolation calibration, press **ISOLATION**, **OMIT ISOLATION** and go to step 19.
14. If you want the isolation calibration, press **Avg**, **AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)
15. Press **Cal**, **RESUME CAL SEQUENCE**.
16. Connect the isolation standards as shown in [Figure 4-11](#).

Figure 4-11. E/O Isolation Calibration Setup



17. Press **ISOLATION**, **FWD ISOL'N**, **ISOL'N STD**. The E/O isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.
18. Press **Avg**, **AVERAGING OFF**, **Cal**, **RESUME CAL SEQUENCE**.
19. Press **ISOLATION DONE**, **DONE: RESP & MATCH**. The frequency response and isolation error coefficients are computed and stored. A corrected trace is displayed and the notation "Cor" appears at the left side of the screen.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

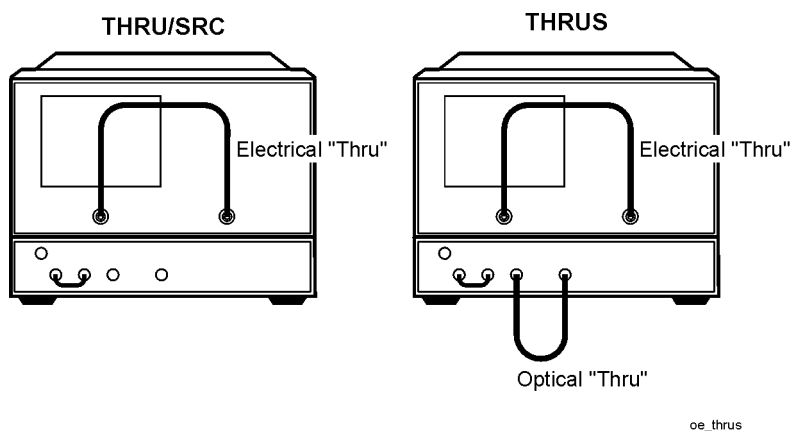
This completes the response and match correction for electrical-to-optical device measurements. You can connect and measure your device under test

O/E Response Calibration

By following this procedure, you can effectively remove the frequency response errors of the test setup for transmission measurements of optical-to-electrical devices.

1. Press **Meas, Trans: O/E (Port 1)**.
2. Select any other settings that you want for the device measurement: power, sweep type, number of points, or IF bandwidth.
3. Press **Cal, CALIBRATE MENU, RESPONSE**.
4. Connect the calibration standards as shown in [Figure 4-12](#). The THRU configuration is recommended.

Figure 4-12. O/E Response Calibration Setup



5. Measure the calibration standards:
 - If you connected both the electrical and optical thru, press **THRU**.
 - If you only connected the electrical thru, press **THRU/SRC**.
6. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The analyzer underlines the softkey that you selected after it finishes the measurement, and computes the error coefficients. Press **DONE: RESPONSE**.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

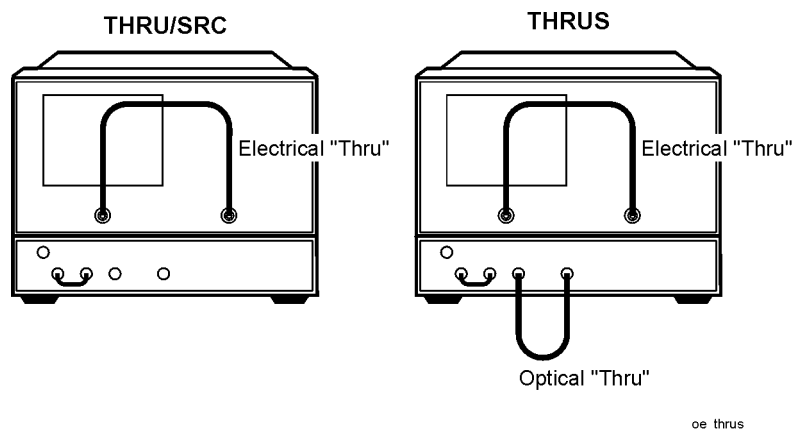
This completes the response correction for optical-to-electrical device measurements. You can connect and measure your device under test.

O/E Response and Isolation Calibration

By following this procedure, you can effectively remove the frequency response and isolation errors of the test setup for transmission measurements or the directivity error in reflection measurements of optical-to-electrical devices with wide dynamic range.

1. Press **Meas, Trans: O/E (Port 1)**.
2. Select any other settings that you want for the device measurement: power, sweep type, number of points, or IF bandwidth.
3. Press **Cal, CALIBRATE MENU, RESPONSE&ISOL'N, RESPONSE**.
4. Connect the calibration standards as shown in [Figure 4-13](#). The THRU configuration is recommended.

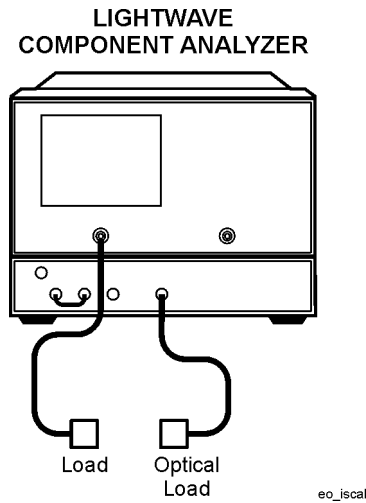
Figure 4-13. O/E Response Calibration Setup



5. Measure the calibration standards:
 - If you connected both the electrical and optical thrus, press **THRU**.
 - If you only connected the electrical thru, press **THRU/SRC**.
6. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. Press **DONE: RESPONSE**.
7. Press **Avg, AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)
8. Press **Cal, RESUME CAL SEQUENCE**.

9. Connect the isolation standards as shown in [Figure 4-14](#).

Figure 4-14. O/E Isolation Calibration Setup



10. Press **ISOL'N STD**. The O/E isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.

11. Press **Avg**, **AVERAGING OFF**, **Cal**, **RESUME CAL SEQUENCE**, **DONE: RESP ISOL'N CAL**. The frequency response and isolation error coefficients are computed and stored. A corrected trace is displayed and the notation "Cor" appears at the left side of the screen.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to "Saving and Recalling Measurements and Data."

12. This completes the response and isolation correction for optical-to-electrical device measurements. You can connect and measure your device under test.

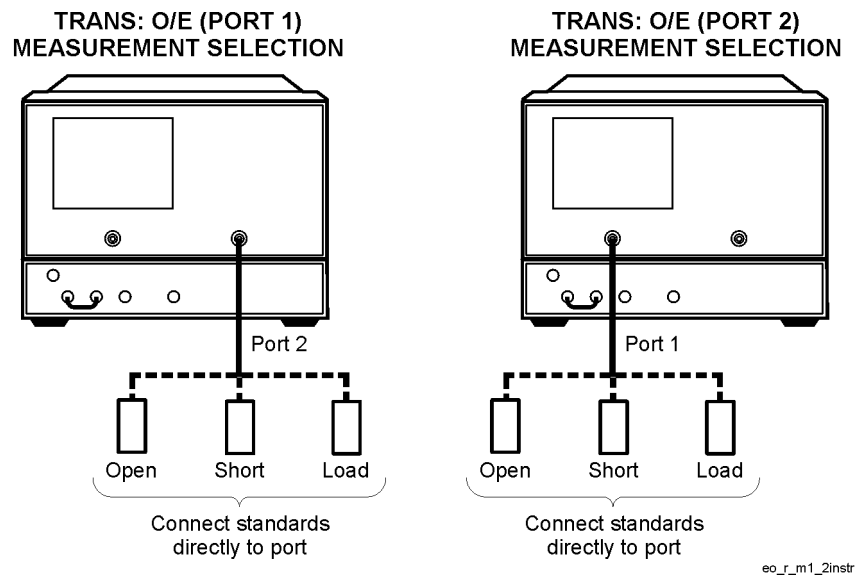
O/E Response and Match Calibration

NOTE The PORT 1 configuration is recommended to achieve the best results.

By following this procedure, you can effectively remove the frequency response and port match errors of the test setup for transmission measurements of optical demodulators.

1. Press **Meas** and select the measurement and electrical test port:
 - If you are using electrical PORT 1, press **Trans: O/E (Port 1)**.
 - If you are using electrical PORT 2, press **Trans: O/E (Port 2)**
2. Press **Avg, IF BW, 100, x1** to reduce the the IF bandwidth (10 Hz IF BW for maximum dynamic range).
3. Select any other settings that you want for the device measurement: power, sweep type, or number of points.
4. Press **Cal, CALIBRATE MENU, RESPONSE & MATCH, REFLECTION**.
5. Connect the open calibration standard directly to the electrical port as shown in [Figure 4-15](#).

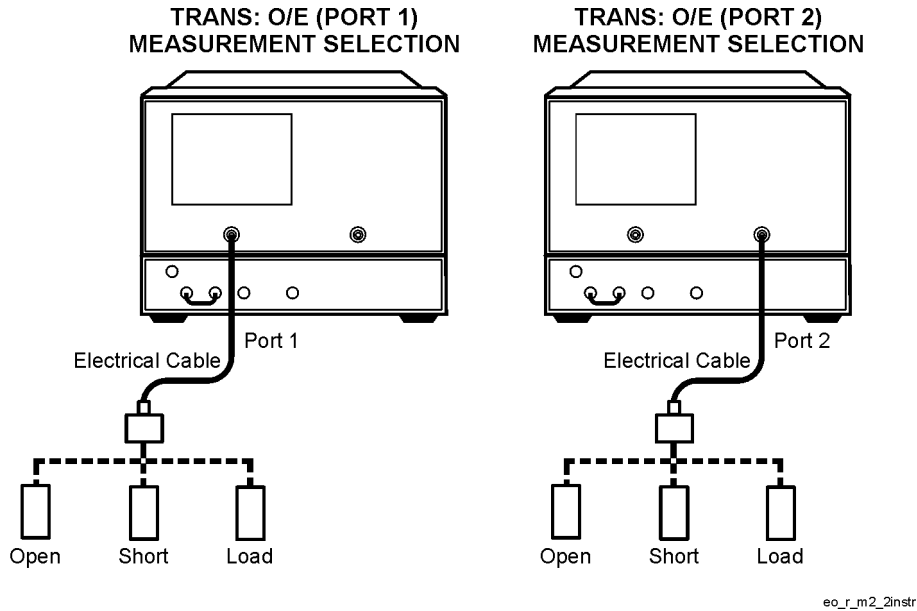
Figure 4-15. Forward Reflection Measurement for O/E Response and Match Calibration



6. Measure the calibration standard by pressing **FORWARD: OPEN**.
7. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The softkey label is then underlined.
8. Repeat steps 5 through 7 to measure the short and the load calibration standards in the forward direction.

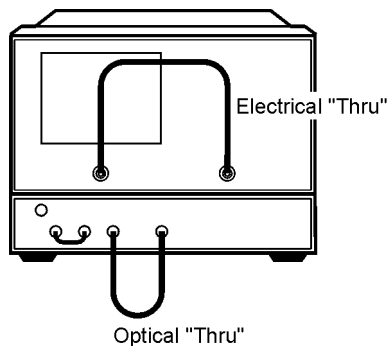
9. Connect the open calibration standard to the end of a cable that is connected to the electrical port as shown in [Figure 4-16](#).

Figure 4-16. Reverse Reflection Measurement for O/E Response and Match Calibration



10. Measure the calibration standard by pressing **REVERSE: OPEN**.
11. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The softkey label is then underlined.
12. Repeat steps 9 through 11 to measure the short and the load calibration standards in the reverse direction.
13. Press **STANDARDS DONE**.
14. Connect the cables, as shown in [Figure 4-17](#).

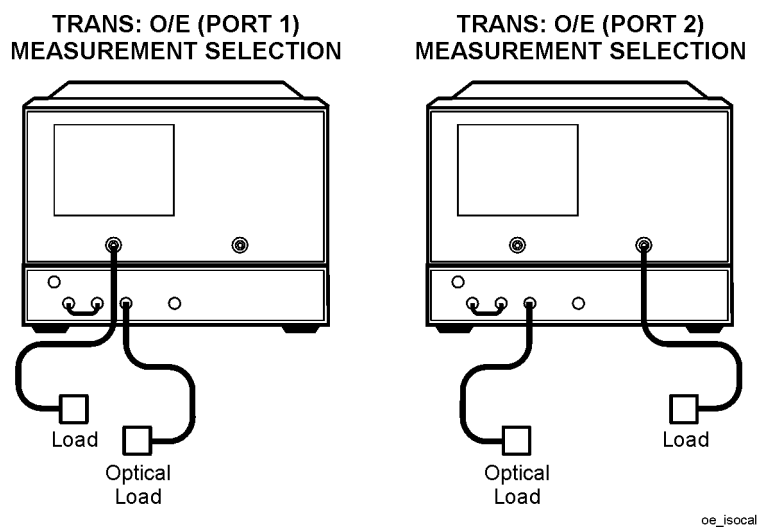
Figure 4-17. Transmission Measurement for O/E Response and Match Calibration



15. Press **TRANSMISSION, FWD. TRANS. THRU**. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement.

16. Press **FWD. MATCH THRU**. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The softkey label is then underlined.
17. Press **STANDARDS DONE**.
18. If you do not want the isolation calibration, press **ISOLATION, OMIT ISOLATION** and go to step 24.
19. If you want the isolation calibration, press **Avg, AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)
20. Press **Cal, RESUME CAL SEQUENCE, ISOLATION**.
21. Connect the isolation standards as shown in [Figure 4-18](#).

Figure 4-18. Isolation Measurement for O/E Response and Match Calibration



22. Press **FWD ISOL'N, ISOL'N STD**. The O/E isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.
23. Press **ISOLATION DONE, Avg, AVERAGING OFF, Cal, RESUME CAL SEQUENCE**.
24. Press **DONE: RESP & MATCH**. The frequency response and isolation error coefficients are computed and stored. A corrected trace is displayed and the notation “Cor” appears at the left side of the screen.

NOTE You can save or store the measurement correction to use for later measurements, that use the same measurement parameters. Refer to “Saving and Recalling Measurements and Data.”

This completes the response and match correction for optical-to-electrical device measurements. You can connect and measure your device under test

Modifying Lightwave Calibration Kits

Characteristics of the optical calibration standard models in the analyzer can be modified to approach more closely the actual standards in use.

Definitions

- **Fresnel Reflection:** The reflection of a portion of incident light at a planar interface between two homogenous media having different refractive indexes.

For calibration standards, this term is used to refer to an optical fiber-air interface. The most common type of Fresnel reflection used for calibration is that formed by an unterminated connector. Theoretically the amount of power reflected at this interface is 3.5%. That is, 3.5% of the optical power is reflected back to the source and 96.5% is transmitted from the cable into air.

- **Reflector:** A device that causes that abrupt reversal of direction of a light beam. Reflection from a smooth surface is termed specular, while reflection from a rough surface is termed diffuse.

The theoretical power reflected is 100%. However, this factor can be adjusted from 0% to 100% to account for any practical reflector being used in the calibration.

- **Thru:** Low loss optical cables that transfer a light beam.

Lightwave Kit Menus

This menu is accessed from the Cal key and leads to an additional menu used to modify the three optical standards.

The modify optical kit menu presents the three optical standards and leads to additional menus containing the specific characteristics for each of the standards.

1. Press **Cal**, **CAL KIT**, **OPT. KIT**, **MODIFY STANDARDS**, and select the optical standard.
 - Press **MODIFY: REFLECTOR** if you want to define the reflector standard.
 - Press **MODIFY: FRESNEL** if you want to define the Fresnel reflection standard.
 - Press **MODIFY: THRU** if you want to define the optical thru standard.
2. Modify the standard model coefficients.
3. Press **STD DONE (DEFINED)**, **DONE MODIFIED**, **SAVE OPT STANDARDS**.

Modifying an Electrical Calibration Kit

Procedure

The following steps are used to modify or define a user kit:

1. Select the predefined kit to be modified. (This is not necessary for defining a new calibration kit.)
2. Define the standards:
 - Define which "type" of standard it is.
 - Define the electrical characteristics (coefficients) of the standard.
3. Specify the class where the standard is to be assigned.
4. Store the modified calibration kit.

Modify Calibration Kit Menu

The **MODIFY []**, softkey in the cal kit menu accesses the modify calibration kit menu. This leads in turn to additional series of menus associated with modifying calibration kits. The following is a description of the softkeys located within this menu:

- **DEFINE STANDARD**, makes the standard number the active function, and brings up the define standard menus. Before selecting a standard, a standard number must be entered. This number (1 to 8) is an arbitrary reference number used to reference standards while specifying a class. The standard numbers for the predefined calibration kits are as follows:

- 1 short (m)
- 2 open (m)
- 3 broadband load
- 4 thru
- 5 sliding load
- 6 lowband load
- 7 short (f)
- 8 open (f)

NOTE Although the numbering sequences are arbitrary, confusion can be minimized by using consistency. However, standard 5 is always a sliding load.

- **SPECIFY CLASS**, leads to the specify class menu. After the standards are modified, use this key to specify a class to group certain standards.

- **LABEL CLASS**, leads to the label class menu, to give the class a meaningful label for future reference.
- **LABEL KIT**, leads to a menu for constructing a label for the user-modified cal kit. If a label is supplied, it will appear as one of the five softkey choices in the select cal kit menu. The approach is similar to defining a display title, except that the kit label is limited to ten characters.
- **TRL/LRM OPTION**, brings up the TRL Option menu.
- **KIT DONE (MODIFIED)**, terminates the calibration kit modification process, after all standards are defined and all classes are specified. Be sure to save the kit with the **SAVE USER KIT**, softkey, if it is to be used later.

Define Standard Menus

Standard definition is the process of mathematically modeling the electrical characteristics (delay, attenuation, and impedance) of each calibration standard. These electrical characteristics (coefficients) can be mathematically derived from the physical dimensions and material of each calibration standard, or from its actual measured response. The parameters of the standards can be listed in [Table 4-2](#).

Table 4-2. Standard Definitions

System $Z_0^a =$ _____ Calibration Kit Label _____

Disk File Name: _____

STANDARD ^b		C0 ^c ×10 ⁻¹⁵ F	C1 ^c ×10 ⁻²⁷ F/Hz	C2 ^c ×10 ⁻³⁶ F/Hz ²	C3 ^c ×10 ⁻⁴⁵ F/Hz ³	FIXED ^d SLIDING or OFFSET	TERM ^e IMPED Ω	OFFSET			FREQ (GHz)		COAX or WG	STND LABEL
NO.	TYPE							DELAY s	Z ₀ Ω	LOSS Ω/s	MIN	MAX		
1														
2														
3														
4														
5														
6														
7														
8														

- Ensure system Z_0 of analyzer is set to this value.
- Open, short, load, delay/thru, or arbitrary impedance.
- Open standard types only.
- Load or arbitrary impedance only.
- Arbitrary impedance only, device terminating impedance.

Each standard must be identified as one of five "types": open, short, load, delay/thru, or arbitrary impedance.

After a standard number is entered, selection of the standard type will present one of five menus for entering the electrical characteristics (model coefficients) corresponding to that standard type, such as **OPEN**. These menus are tailored to the current type, so that only characteristics applicable to the standard type can be modified.

The following is a description of the softkeys located within the define standard menu:

- **OPEN**, defines the standard type as an open, used for calibrating reflection measurements. Opens are assigned a terminal impedance of infinite ohms, but delay and loss offsets may still be added. Pressing this key also brings up a menu for defining the open, including its capacitance.

As a reflection standard, an open termination offers the advantage of broadband frequency coverage. At RF and microwave frequencies, however, an open rarely has perfect reflection characteristics because fringing capacitance effects cause phase shift that varies with frequency. This can be observed in measuring an open termination after calibration, when an arc in the lower right circumference of the Smith chart indicates capacitive reactance. These effects are impossible to eliminate, but the calibration kit models include the open termination capacitance at all frequencies for compatible calibration kits. The capacitance model is a cubic polynomial, as a function of frequency, where the polynomial coefficients are user-definable. The capacitance model equation is:

$$C = (C0) + (C1 \times F) + (C2 \times F^2) + (C3 \times F^3)$$

where F is the measurement frequency.

The terms in the equation are defined with the specify open menu as follows:

- **C0**, allows you to enter the C0 term, which is the constant term of the cubic polynomial and is scaled by 10^{-15} Farads.
- **C1**, allows you to enter the C1 term, expressed in F/Hz (Farads/Hz) and scaled by 10^{-27} .
- **C2**, allows you to enter the C2 term, expressed in F/Hz² and scaled by 10^{-36} .
- **C3**, allows you to enter the C3 term, expressed in F/Hz³ and scaled by 10^{-45} .
- **SHORT**, defines the standard type as a short, for calibrating reflection measurements. Shorts are assigned a terminal impedance of 0 ohms, but delay and loss offsets may still be added.
- **LOAD**, defines the standard type as a load (termination). Loads are assigned a terminal impedance equal to the system characteristic impedance Z0, but delay and loss offsets may still be added. If the load impedance is not Z0, use the arbitrary impedance standard definition.
 - **FIXED**, defines the load as a fixed (not sliding) load.
 - **SLIDING**, defines the load as a sliding load. When such a load is measured during calibration, the analyzer will prompt for several load positions, and calculate the ideal load value from it.
 - **OFFSET**, defines the load as being offset.

- **DELAY/THRU**, defines the standard type as a transmission line of specified length, for calibrating transmission measurements.
- **ARBITRARY IMPEDANCE**, defines the standard type to be a load, but with an arbitrary impedance (different from system Z_0).
 - **TERMINAL IMPEDANCE**, allows you to specify the (arbitrary) impedance of the standard, in ohms.
 - **FIXED**, defines the load as a fixed (not sliding) load.
 - **SLIDING**, defines the load as a sliding load. When such a load is measured during calibration, the analyzer will prompt for several load positions, and calculate the ideal load value from it.

Any standard type can be further defined with offsets in delay, loss, and standard impedance; assigned minimum or maximum frequencies over which the standard applies, and defined as coax or waveguide.

- The **SPECIFY OFFSET**, softkey accesses the specify offset menu (described next).
- The **LABEL STD**, softkey allows you to define a distinct label for each standard, so that the analyzer can prompt you with explicit standard labels during calibration (such as **SHORT**). The function is similar to defining a display title, except that the label is limited to ten characters.

After each standard is defined, including offsets, the **STD DONE (DEFINED)**, softkey will terminate the standard definition.

Specify Offset Menu

The specify offset menu allows additional specifications for a user-defined standard. Features specified in this menu are common to all five types of standards.

Offsets may be specified with any standard type. This means defining a uniform length of transmission line to exist between the standard being defined and the actual measurement plane. (Example: a waveguide short circuit terminator, offset by a short length of waveguide.) For reflection standards, the offset is assumed to be between the measurement plane and the terminating element of the standard (one-way only). For transmission standards, the offset is assumed to exist between the two reference planes (in effect, the offset is the thru). For both reflection and transmission, the offset is entered as a one-way offset. Three characteristics of the offset can be defined: its delay (length), loss, and impedance.

In addition, the frequency range over which a particular standard is valid can be defined with a minimum and maximum frequency. This is particularly important for a waveguide standard, since the minimum frequency is used to define the waveguide cutoff frequency. Note that several band-limited standards can together be defined as the same "class" (see ["Specify Class Menu" on page 4-35](#)). Then, if a measurement calibration is performed over a frequency range exceeding a single standard, additional standards can be used for each portion of the range.

Lastly, the standard must be defined as either coaxial or waveguide. If it is waveguide, dispersion effects are calculated automatically and included in the standard model.

The following is a description of the softkeys located within the specify offset menu:

- **OFFSET DELAY**, allows you to specify the one-way electrical delay from the measurement (reference) plane to the standard, in seconds (s). (In a transmission standard, offset delay is the delay from plane to plane.) Delay can be calculated from the precise physical length of the offset, the permittivity constant of the medium, and the speed of light.

In coax, group delay is considered constant. In waveguide, however, group delay is dispersive, that is, it changes significantly as a function of frequency. Hence, for a waveguide standard, offset delay must be defined as though it were a TEM wave (without dispersion).

- **OFFSET LOSS**, allows you to specify energy loss, due to skin effect, along a one-way length of coax offset. The value of loss is entered as ohms/nanosecond (or Giga ohms/second) at 1 GHz. (Such losses are negligible in waveguide, so enter 0 as the loss offset.)
- **OFFSET Z0**, allows you to specify the characteristic impedance of the coax offset. (Note: This is not the impedance of the standard itself.) For waveguide, the offset impedance as well as the system Z0 must always be set to 1Ω .
- **MINIMUM FREQUENCY**, allows you to define the lowest frequency at which the standard can be used during measurement calibration. In waveguide, this must be the lower cutoff frequency of the standard, so that the analyzer can calculate dispersive effects correctly (see **OFFSET DELAY**).
- **MAXIMUM FREQUENCY**, allows you to define the highest frequency at which the standard can be used during measurement calibration. In waveguide, this is normally the upper cutoff frequency of the standard.
- **COAX**, defines the standard (and the offset) as coaxial. This causes the analyzer to assume linear phase response in any offsets.
- **WAVEGUIDE**, defines the standard (and the offset) as rectangular waveguide. This causes the analyzer to assume a dispersive delay (see **OFFSET DELAY**).

Label Standard Menu

This menu allows you to label (reference) individual standards during the menu-driven measurement calibration sequence. The labels are user-definable using a character set shown on the display that includes letters, numbers, and some symbols, and they may be up to ten characters long. The analyzer will prompt you to connect standards using these labels, so they should be meaningful to you, and distinct for each standard.

By convention, when sexed connector standards are labeled male (m) or female (f), the designation refers to the test port connector sex, not the connector sex of the standard.

Specify Class Menu

Once a standard has been defined, it must be assigned to a standard "class." This is a group of from one to seven standards that is required to calibrate for a single error term. The standards within a single class can be assigned to the locations listed in [Table 4-3](#) according to their standard reference numbers.

A class often consists of a single standard, but may be composed of more than one standard if band-limited standards are used. For example, if there were two load standards—a fixed load for low frequencies, and a sliding load for high frequencies—then that class would have two standard. The number of standard classes required depends on the type of calibration being performed, and is identical to the number of error terms corrected. A response calibration requires only one class, and the standards for that class may include an open, or short, or thru. A 1-port calibration requires three classes. A full 2-port calibration requires 10 classes, not including two for isolation.

The number of standards that can be assigned to a given class may vary from none (class not used) to one (simplest class) to seven. When a certain class of standards is required during calibration, the analyzer will display the labels for all the standards in that class (except when the class consists of a single standard). This does not, however, mean that all standards in a class must be measured during calibration. Unless band-limited standards are used, only a single standard per class is required.

Table 4-3. Standard Class Assignments

Calibration Kit Label: _____

Disk File Name: _____

Class	Standard Reference Numbers								Standard Class Label
	1	2	3	4	5	6	7	8	
S ₁₁ A									
S ₁₁ B									
S ₁₁ C									
S ₂₂ A									
S ₂₂ B									
S ₂₂ C									
Forward Transmission									
Reverse Transmission									
Forward Match									
Reverse Match									
Response									
Response and Isolation									
TRL thru									
TRL reflect									
TRL line or match									

NOTE It is often simpler to keep the number of standards per class to the bare minimum needed (often one) to avoid confusion during calibration.

Each class can be given a user-definable label as described under label class menus. Standards are assigned to a class simply by entering the standard's reference number (established while defining a standard) under a particular class. The following is a description of the softkeys located within the specify class menu:

- **S11A**, allows you to enter the standard numbers for the first class required for an S₁₁ 1-port calibration. (For default calibration kits, this is the open.)

- **S11B**, allows you to enter the standard numbers for the second class required for an S_{11} 1-port calibration. (For default calibration kits, this is the short.)
- **S11C**, allows you to enter the standard numbers for the third class required for an S_{11} 1-port calibration. (For default calibration kits, this is the load.)
- **S22A**, allows you to enter the standard numbers for the first class required for an S_{22} 1-port calibration. (For default calibration kits, this is the open.)
- **S22B**, allows you to enter the standard numbers for the second class required for an S_{22} 1-port calibration. (For default calibration kits, this is the short.)
- **S22C**, allows you to enter the standard numbers for the third class required for an S_{22} 1-port calibration. (For default calibration kits, this is the load.)
- **FWD TRANS**, allows you to enter the standard numbers for the forward transmission thru calibration. (For default calibration kits, this is the thru.)
- **REV TRANS**, allows you to enter the standard numbers for the reverse transmission (thru) calibration. (For default calibration kits, this is the thru.)
- **FWD MATCH**, allows you to enter the standard numbers for the forward match (thru) calibration. (For default calibration kits, this is the thru.)
- **REV MATCH**, allows you to enter the standard numbers for the reverse match (thru) calibration. (For default calibration kits, this is the thru.)
- **RESPONSE**, allows you to enter the standard numbers for a response calibration. This calibration corrects for frequency response in either reflection or transmission measurements, depending on the parameter being measured when a calibration is performed. (For default kits, the standard is either the open or short for reflection measurements, or the thru for transmission measurements.)
- **RESPONSE & ISOL'N**, allows you to enter the standard numbers for a response & isolation calibration. This calibration corrects for frequency response and directivity in reflection measurements, or frequency response and isolation in transmission measurements.
- **TRL THRU**, allows you to enter the standard numbers for a TRL thru calibration.
- **TRL REFLECT**, allows you to enter the standard numbers for a TRL reflect calibration.
- **TRL LINE OR MATCH**, allows you to enter the standard numbers for a TRL line or match calibration.

Label Class Menu

The label class menus are used to define meaningful labels for the calibration classes. These then become softkey labels during a measurement calibration. Labels can be up to ten characters long.

Label Kit Menu

This **LABEL KIT**, softkey within the modify cal kit menu, accesses this menu. It is identical to the label class menu and the label standard menu previously described. It allows definition of a label up to eight characters long.

After a new calibration kit has been defined, be sure to specify a label for it. Choose a label that describes the connector type of the calibration devices. This label will then appear in the **CAL KIT []**, softkey label in the correction menu and the **MODIFY []**, label in the select cal kit menu. It will be saved with calibration sets.

Saving Modified Calibration Kits to a Disk

The calibration kit, along with any calibration data and other instrument state information, can be saved to an ISTATE file on a floppy disk. To save a modified calibration kit with an instrument state, press:

Cal, CAL KIT, SELECT CAL KIT, USER KIT

Save/Recall, SAVE STATE

Verify Performance

Once a measurement calibration has been generated with a user-defined calibration kit, its performance should be checked before making device measurements. To check the accuracy that can be obtained using the new calibration kit, a device with a well-defined frequency response (preferably unlike any of the standards used) should be measured. The verification device must not be one of the calibration standards: measurement of one of these standards is merely a measure of repeatability.

To achieve a more complete verification you can use the N1011A Lightwave Verification Kit. This kit includes an accurately known verification standard and corresponding response data to compare with the analyzer measurement data.

Verifying Measurement Accuracy

Verifying Measurement Accuracy

In this chapter, you can find a procedure for verifying the correct operation of the analyzer. During this procedure, the analyzer measures a device and compares the results to the device data, which is included in the required verification kit.

Lightwave Verification Procedure

To get the best indication of instrument changes over time, print the results from the first test and archive them to compare with all subsequent tests. The verification routine sets the analyzer to the following conditions:

- Full band sweep (50 MHz to 20.05 GHz)
- 401 points
- 30 Hz IF bandwidth
- 0.5 % Smoothing
- 8 Sweep Averages

WARNING The verification routine saves data to Memory 2, overwriting any data previously stored in Memory 2.

Required Equipment

- 8703B Lightwave Component Analyzer
- Verification Kit (N1011A)
- Floppy Disk (included in verification kit)

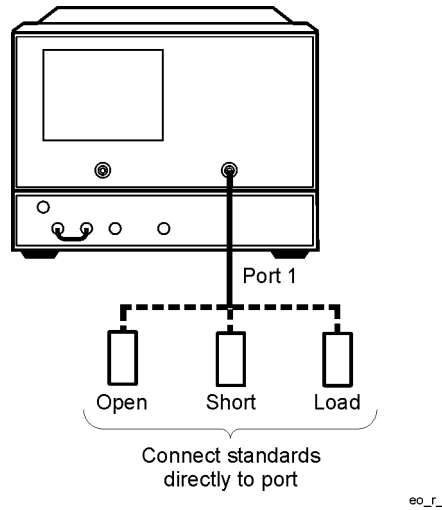
1. Press **System, Service Menu, Lightwave Service Menu, Verify Instrument.**

The conditions set for the verification are as follows:

NOTE Do not change the conditions during the verification procedure.

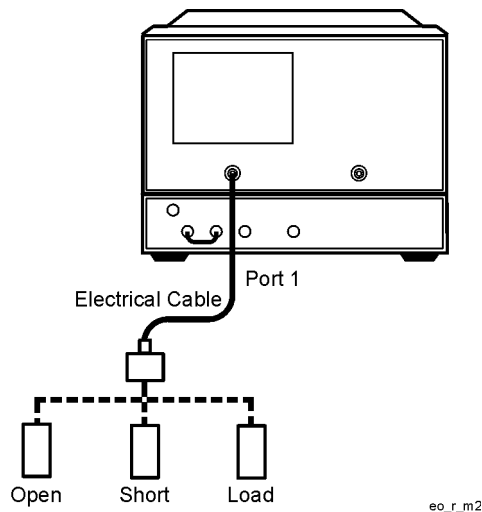
2. Press **Verify ON, O/E Cal** and perform an O/E Response and Match Calibration.
3. Press **Meas, Trans: O/E, Trans: O/E (Port 1).**
4. Press **Cal, CALIBRATE MENU, RESPONSE & MATCH, REFLECTION.**
5. Connect the open calibration standard directly to the electrical port as shown in [Figure 5-1](#).

Figure 5-1. Forward Reflection Measurement for O/E Response and Match Calibration



6. Measure the calibration standard by pressing **FORWARD: OPEN**.
7. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The softkey label is then underlined.
8. Repeat steps 5 through 7 to measure the short and the load calibration standards in the forward direction.
9. Connect the open calibration standard to the end of a cable that is connected to the electrical port as shown in [Figure 5-2](#).

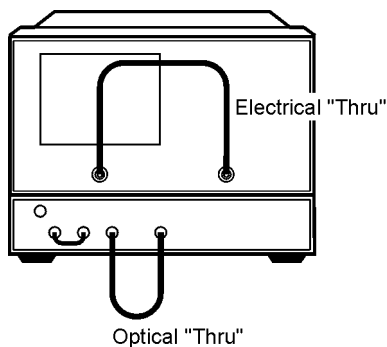
Figure 5-2. Reverse Reflection Measurement for O/E Response and Match Calibration



10. Measure the calibration standard by pressing **REVERSE: OPEN**.
11. The analyzer displays “WAIT - MEASURING CAL STANDARD” during the standard measurement. The softkey label is then underlined.

12. Repeat steps 9 through 11 to measure the short and the load calibration standards in the reverse direction.
13. Press **STANDARDS DONE**.
14. Connect the cables, as shown in [Figure 5-3](#).

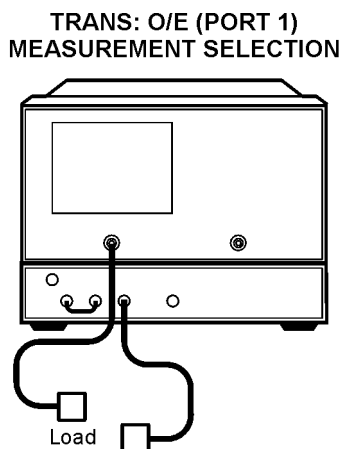
Figure 5-3. Transmission Measurement for O/E Response and Match Calibration



oe_r_m3

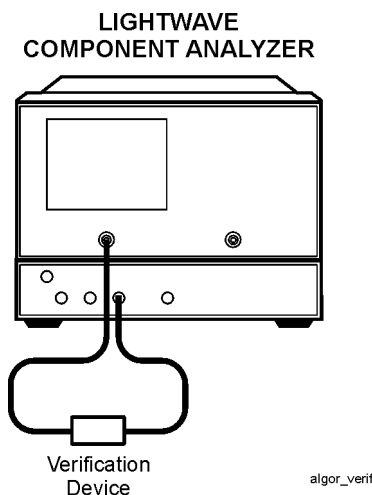
15. Press **TRANSMISSION, FWD. TRANS. THRU**. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The softkey label is then underlined.
16. Press **FWD. MATCH THRU**. The analyzer displays "WAIT - MEASURING CAL STANDARD" during the standard measurement. The softkey label is then underlined.
17. Press **STANDARDS DONE**.
18. If you do not want the isolation calibration, press **ISOLATION, OMIT ISOLATION** and go to step 24.
19. If you want the isolation calibration, press **Avg, AVERAGING ON**. (For greater dynamic range use an averaging factor of 32.)
20. Press **Cal, RESUME CAL SEQUENCE, ISOLATION**.
21. Connect the isolation standards as shown in [Figure 5-4](#).

Figure 5-4. Isolation Measurement for O/E Response and Match Calibration



22. Press **FWD ISOL'N, ISOL'N STD**. The O/E isolation data is measured and averaged over 16 sweeps (or the number selected). The softkey label is then underlined.
23. Press **ISOLATION DONE, Avg, AVERAGING OFF, Cal, RESUME CAL SEQUENCE**.
24. Press **DONE: RESP & MATCH**.
25. Insert the floppy disk that has the verification device data into the analyzer disk drive.
26. Press **Load Disk**.
27. Connect the verification device between the analyzer lightwave source and PORT 1.

Figure 5-5. Setup for Verification Device Measurement



28. Press **Start Verify**. This test compares the measured data with the data from the floppy disk. The message "PASS" or "FAIL" will appear on the analyzer.
29. Press **View Deviation** to toggle between seeing the data as a comparison or as the deviation with the limits:
 - Comparison compares the trace with the verification device data at a point-by-point basis. If one point fails in the comparison, the analyzer will fail the verification test.

The DC responsivity is not verified. The DC offset is calculated as the difference between the average value of the measured data (20 points) and the average value of the verification device data (20 points).

If the Analyzer Does Not Pass the Verification Test

Clean all of the connectors and cable ends and repeat the verification procedure. If the analyzer continues to fail the test, contact your nearest Agilent Technologies office or sales representative. A list of Agilent Technologies sales and service offices is provided at the end of this document.

Agilent Technologies Service Offices

Before returning an instrument for service, call the Agilent Technologies Instrument Support Center at (800) 403-0801, visit the Test and Measurement Web Sites by Country page at <http://www.tm.agilent.com/tmo/country/English/index.html>, or call one of the numbers listed below.

Table 5-1. Agilent Technologies Service Numbers

Austria	01/25125-7171
Belgium	32-2-778.37.71
Brazil	(11) 7297-8600
China	86 10 6261 3819
Denmark	45 99 12 88
Finland	358-10-855-2360
France	01.69.82.66.66
Germany	0180/524-6330
India	080-34 35788
Italy	+39 02 9212 2701
Ireland	01 615 8222
Japan	(81)-426-56-7832
Korea	82/2-3770-0419
Mexico	(5) 258-4826
Netherlands	020-547 6463
Norway	22 73 57 59
Russia	+7-095-797-3930
Spain	(34/91) 631 1213
Sweden	08-5064 8700
Switzerland	(01) 735 7200
United Kingdom	01 344 366666
United States and Canada	(800) 403-0801

6

- “Saving and Recalling Measurements and Data” on page 6-2
- “Saving an Instrument State” on page 6-5
- “Saving Measurement Results” on page 6-6
- “Re-Saving an Instrument State” on page 6-18
- “Deleting a File” on page 6-19
- “Renaming a File” on page 6-20
- “Recalling a File” on page 6-21
- “Formatting a Disk” on page 6-22
- “Solving Problems with Saving or Recalling Files” on page 6-23

Saving and Recalling Measurements and Data

Saving and Recalling Measurements and Data

This chapter describes the various places that you can store measurement setups and data, and why you may want to select a particular storage medium. You will also find procedures in this chapter for saving and recalling measurement settings and data.

Saving and Recalling Instrument States

Places Where You Can Save

- analyzer internal memory
- floppy disk using the analyzer's internal disk drive
- floppy disk using an external disk drive
- IBM compatible personal computer using GPIB mnemonics

What You Can Save to the Analyzer's Internal Memory

The number of registers that the analyzer allows you to save depends on the size of associated error-correction sets, and memory traces. However, the maximum number of registers that can be saved to internal memory is 31. Refer to the "Preset State and Memory Allocation" chapter of the reference guide for further information.

You can save instrument states in the analyzer internal memory, along with the following list of analyzer settings. The default file names are REG<01-31>.

- error-corrections on channels 1 and 2
- displayed memory trace
- print/plot definitions
- measurement setup
 - frequency range
 - number of points
 - sweep time
 - output power
 - sweep type
 - measurement parameter

NOTE When the ac line power is switched off, the internal non-volatile memory is retained by a battery. Refer to "Specifications and Characteristics" in the reference guide for data retention times.

What You Can Save to a Floppy Disk

You can save an instrument state and measurement results to a disk. The default file names are FILE n , where n gets incremented by one each time a file with a default name is added to the directory. The default file names for data-only files are DATA y D z (DATA y .D z for DOS), where y is incremented by one each time a file with a default name is added to the directory. The z is the channel where the measurement was made. When you save a file to disk, you can choose to save some or all of the following:

- all settings listed for internal memory
- active error-correction for the active channel only
- displayed measurement data trace
- displayed user graphics
- data only
- HPGL plots

What You Can Save to a Computer

Instrument states can be saved to and recalled from an external computer (system controller) using GPIB mnemonics. For more information about the specific analyzer settings that can be saved, refer to the output commands located in the "Command Reference" chapter of the programmer's guide. For an example program, refer to the "Programming Examples" chapter of the programmer's guide.

Saving an Instrument State

1. Press **Save/Recall, SELECT DISK**, and select one of the storage devices:
 - INTERNAL MEMORY**
 - INTERNAL DISK**
 - EXTERNAL DISK**, and configure as follows:
 - a. Connect an external disk drive to the analyzer's GPIB connector.
 - b. Press **Local, DISK UNIT NUMBER**, and enter the drive where your disk is located, followed by **x1**.
 - c. If your storage disk is partitioned, press **VOLUME NUMBER**, and enter the volume number where you want to store the instrument state file.
 - d. Press **SET ADDRESSES, ADDRESS: DISK**.
 - e. Enter the GPIB address of the peripheral, if the default address is incorrect (default = 00). Follow the entry by pressing **x1**.
 - f. Press **Local**, and select one of the following:
 - **SYSTEM CONTROLLER**, allows the analyzer to control peripherals directly.
 - **TALKER/LISTENER**, allows the computer controller to be involved in all peripheral access operations.
 - **USE PASS CONTROL**, allows you to control the analyzer over GPIB and also allows the analyzer to take or pass control.
2. Press **Save/Recall, SAVE STATE**.

Although one file is shown to represent an instrument state on the analyzer display, each instrument state is composed of numerous files (which can be viewed on a PC).

NOTE If you have saved enough files that you have used all the default names (FILE00 – FILE31 for disk files, or REG1 – REG31 for memory files), you must do one of the following to save more states:

- use another disk
- rename an existing file to make a default name available
- re-save a file/register
- delete an existing file/register

Saving Measurement Results

Instrument states combined with measurements results can only be saved to disk. Files that contain data-only, and the various save options available under the **DEFINE DISK-SAVE** key, are also only valid for disk saves.

The analyzer stores data in arrays along the processing flow of numerical data, from IF detection to display. These arrays are points in the flow path where data is accessible, usually via GPIB. You can choose from three different arrays which vary in modification flexibility when they are recalled.

- raw data
- data (raw data with error-correction applied) if correction is on; otherwise, raw data
- format (data processed to the display format)

If you choose to save the raw data array, you will have the most flexibility in modifying the recalled measurement (including the ability to view all four S-parameters). This is because the raw data array has the least amount of processing associated with it. Conversely, if you choose to save the format array, your modification of the recalled measurement will be limited by all the processes that are associated with that measurement result. However, the format array is appropriate if you want to retrieve data traces that look like the currently displayed data.

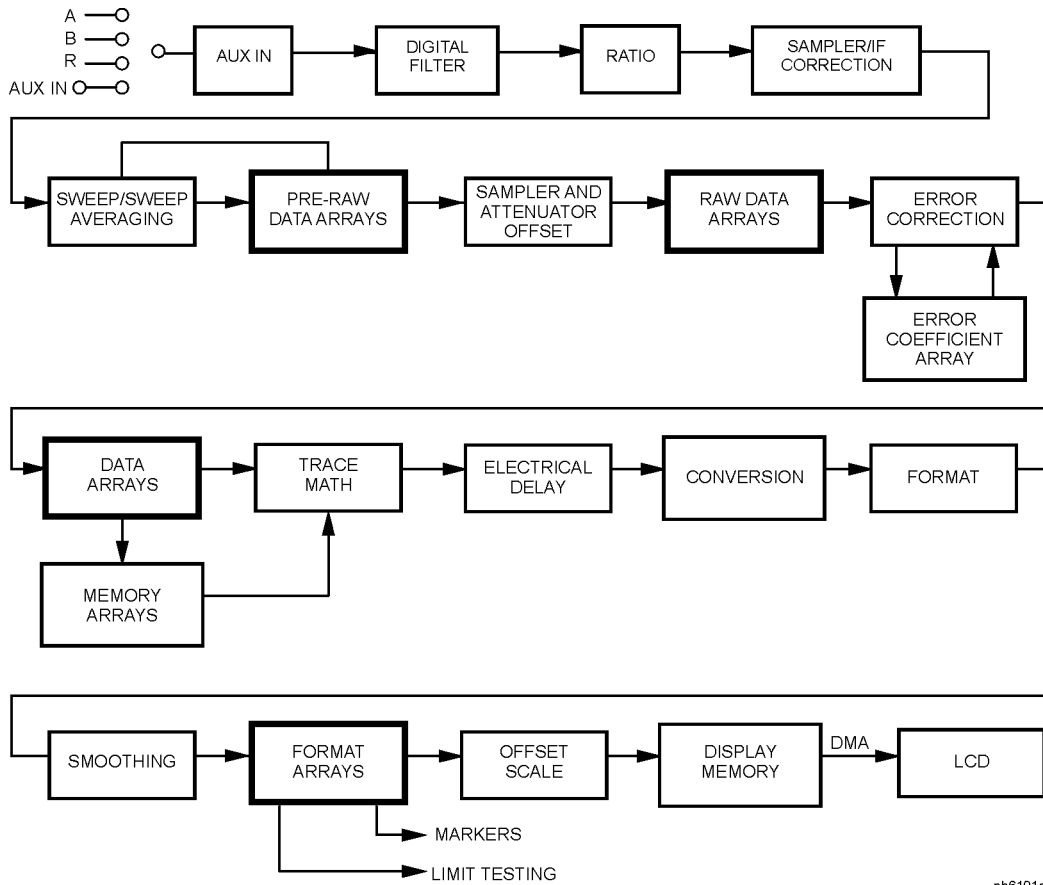
Define Save	Modification Flexibility During Recall
Raw Data Array	Most
Data Array	Medium
Format Array	Least

You can also save data-only. This is saved to disk with default file names

DATA00D1 to DATA31D1 for channel 1
DATA00D2 to DATA31D2 for channel 2
DATA00D3 to DATA31D3 for channel 3
DATA00D4 to DATA31D4 for channel 4

However, these files are not instrument states and cannot be recalled.

Figure 6-1. Data Processing Flow Diagram



NOTE If the analyzer has an active two-port measurement calibration, all four S-parameters will be saved with the measurement results. All four S-parameters may be viewed if the raw data array has been saved.

1. If you want to title the displayed measurement, refer to [“Titling the Displayed Measurement”](#) on page 7-32.
2. Press **Save/Recall, SELECT DISK**.
3. Choose one of the following disk drives:
 - **INTERNAL DISK**
 - **EXTERNAL DISK**, (If necessary, refer to the external disk setup procedure in [“Saving an Instrument State”](#) on page 6-5.)
4. Press **Save/Recall, DEFINE DISK-SAVE**.

5. Define the save by selecting from the following choices:

- **DATA ARRAY ON**
- **RAW ARRAY ON**
- **FORMAT ARY ON**

If you select **DATA ARRAY ON**, **RAW ARRAY ON**, or **FORMAT ARY ON**, the data is stored to disk in IEEE-64 bit real format (for LIF disks), and 32 bit PC format for DOS disks. This makes the DOS data files half the size of the LIF files.

NOTE Selecting **DATA ARRAY ON**, may store data to disk in the S2P ASCII data format. See [“ASCII Data Formats” on page 6-8](#).

GRAPHICS ON

If you select **GRAPHICS ON**, the user graphics area is saved. (Refer to the programmer's guide for information on using display graphics.) The measurement display is not saved with this selection. (Refer to [“If You Are Plotting Measurement Results to a Disk Drive” on page 7-13](#) to plot measurement displays to disk.)

DATA ONLY ON,

If **DATA ONLY ON**, data array is saved along with any other selected array, the instrument state is *not* saved, and therefore, *cannot be recalled*.

6. Choose the type of format you want:

- Choose **SAVE USING BINARY**, for all applications except CITIfile, S2P, or CAE applications.
- Choose **SAVE USING ASCII**, for CITIfile, S2P, and CAE applications or when you want to import the information into a spread sheet format.

7. Press **RETURN, SAVE STATE**.

ASCII Data Formats

CITIfile

CITIfile (Common Instrumentation Transfer and Interchange file) is an ASCII data format that is useful when exchanging data between different computers and instruments. CITIfiles are always saved when the ASCII format has been selected as shown:

Save/Recall

DEFINE DISK-SAVE

Select one of the following choices:

- **DATA ARRAY ON**
 - **DATA ONLY ON**
 - **RAW ARRAY ON**
 - **FORMAT ARY ON**
- SAVE USING ASCII**
RETURN
SAVE STATE

If **DATA ARRAY ON**, or **DATA ONLY ON**, or **FORMAT ARY ON**, is selected, a CITIfile is saved for each displayed channel with the suffix letter “D”, or “F”, followed by a number. The number following “D” and “F” files is the channel number. When **RAW ARRAY ON**, is selected, an “r1” file is saved for channel 1/channel 3, and an “r5” file is saved for channel 2/channel 4. For more information on the CITIFile data format as well as a list of CITIFile keywords, refer to the “Understanding the CITIFile Data Format” chapter in the reference guide.

S2P Data Format

This format creates component data files that describe frequency dependent linear network parameters for 2 port components. These files are assigned a file name with the suffix "S" and are only output (that is, they cannot be read in by the analyzers).

Up to two S2P files are saved: S1 for channel 1, and S2 for channel 2. S2P files are not stored for channel 3 or channel 4 because the data would be redundant. Each S2P file contains all four S-parameter data.

An S2P file is only output when the all of following conditions are met:

- a full two-port or TRL two-port correction is turned on
- **DATA ARRAY ON**, or **DATA ONLY ON**, is selected using **DEFINE DISK SAVE**
- **SAVE USING ASCII**, is selected

Error-corrected data CITI files are always saved along with S2P files.

The template for component data files is as follows:

```
! comment line
# <frequency units> <parameter> <format> <Rn>
<data line>
..
<data line>
where
```

! indicates that all following on this line is a comment

indicates that entries following on this line are parameters that are being specified

frequency units GHz, MHz, kHz, Hz

parameter	S for S-parameters
format	DB for dB magnitude and angle in degrees MA for linear magnitude and angle in degrees RI for real and imaginary pair
Rn	the reference impedance in ohms for the analyzer making the measurement (R 50 or R 75)

The "format" choice is selected by the current selection under the FORMAT menu. To select the DB format, the FORMAT must be LOG MAG. For MA, the FORMAT must be LIN MAG (unlike CITIfile), and all other FORMAT selections will output RI data.

The S2P data will always represent the format array data, including effects of electrical delay and port extensions. A CITIfile will be saved at the same time. To be consistent with previous versions, the CITIfile data saved will represent the DATA array (corrected data) without effects of electrical delay or port extensions.

CAUTION Using the smoothing feature may result in invalid S2P data. Avoid using this function when saving S2P files.

Here is an S2P example file for an 11 point measurement of a 20 dB attenuator:

```
# HZ S DB R 50
! Lightwave Component Analyzer HP8703B.0611
50000000 -56.74 15.178 -20.219 -2.0132 -20.15 -1.6658 -36.188 -123.52
250000000 -53.015 1.7331 -20.373 -10.241 -20.377 -10.029 -33.974 -40.215
450000000 -52.094 5.8173 -20.391 -18.555 -20.387 -17.96 -31.287 61.778
650000000 -51.758 8.02 -20.189 -26.18 -20.112 -26.061 -29.427 153.37
850000000 -50.95 11.472 -20.163 -34.743 -20.198 -34.195 -24.719 -137.83
1050000000 -50.235 9.3562 -20.178 -42.682 -20.19 -42.289 -25.102 -81.096
1250000000 -49.883 9.2574 -20.142 -50.854 -20.223 -50.407 -27.582 -25.509
1450000000 -48.477 5.9944 -20.201 -58.917 -20.21 -58.436 -33.828 35.237
1650000000 -48.462 3.5156 -20.161 -67.008 -20.188 -66.587 -44.184 62.912
1850000000 -47.503 1840 -20.15 -74.862 -20.208 -74.616 -36.893 35.384
2050000000 -46.938 -5.6538 -20.167 -83.048 -20.256 -82.874 -30.385 74.001
```


Saving in Textual (CSV) Form

Textual measurement results can be saved in a comma-separated value (CSV) format and imported into a spreadsheet application. Additional information is also saved as a preamble to the measurement results. The saved information includes:

- Analyzer model number and firmware version
- Date the file was saved
- Type of measurement being done
- Start and stop frequencies
- Sweep time
- Port power
- IF bandwidth
- Channel number
- Number of points
- Format
- The frequency (or time) and the real and imaginary measurement values for each of points measured

1. Press **Save/Recall, SAVE FILE FORMATS**.
2. Make sure that **TEXT FMT [CSV]** is displayed.
3. Make sure that **FILETYPE: TEXT** is underlined. If it is not underlined, press the softkey so that TEXT is underlined.
4. Insert a 3.5-inch floppy disk in the analyzer's disk drive.
5. Press **SAVE FILE**, to save the display information as text in the CSV format.

The text file may be retrieved from the floppy disk on personal computer and can be imported into an application that accepts text in the comma-separated value format, such as a spreadsheet.

How the Analyzer Names These Files Sequentially

When text files are saved, the analyzer generates the file names automatically in the following format:

txtcss.csv

where txt is a constant that indicates that this is a text file,
:

c is the indicator of the channel (1–4) on which the measurement data was taken.

ss is a 2-digit, sequential indicator of the measurement (file index number). The file index number may be numbered from 00–31. As the next measurement is taken, the file index number is incremented.

If all four channels are making measurements and a save is performed, there will be four channel numbers that share the same file index number. For example, the files would be named txt100.csv, txt200.csv, txt300.csv, and txt400.csv. If a measurement does not include all four channels, unused channel-file index numbers will not be used by the next measurement. However, if all of the files that share a file index number are erased, that file index number will be re-used.

csv is the file format, comma-separated value in this case.

Saving in Graphical (JPEG) Form

Graphical measurement results can be saved in JPEG format and used as an illustration in a text editor or desktop publishing application.¹

1. Press **Save/Recall, SAVE FILE FORMATS**.
2. Make sure that **GRAPH FMT [JPG]** is displayed.
3. Make sure that **FILETYPE: GRAPHIC** is underlined. If it is not underlined, press the softkey so that GRAPHIC is underlined.
4. Insert a 3.5-inch floppy disk in the analyzer's disk drive.
5. Press **SAVE FILE**, to save the display as a graphic in the JPEG format.

The graphic file may be retrieved from the floppy disk on personal computer and can be imported into an application that accepts graphics in the JPEG format.

NOTE When saving measurement results graphically, make sure that no onscreen measurement data is displayed as white. Since media color is often white, any measurement data printed using white will not be visible.

You may change the analyzer to the factory default color settings by pressing **Copy, DEFINE PRINT, DEFAULT PRNT SETUP**, to correct this problem.

However, to maintain your current color settings (except white), check the measurement color settings by pressing **Copy, DEFINE PRINT, PRINT COLORS**. Press **MORE**, to check the remaining measurement colors. To modify any of the measurement colors, select the measurement and then choose another color from the list of colors that is displayed.

1. The analyzer firmware is based in part on the work of the Independent JPEG Group.

Instrument State Files

When an instrument state is saved to a floppy disk, some or all of the following files may be produced. This depends upon which arrays are selected under the **DEFINE SAVE STATE**, softkey menu, and whether the selected save format is BINARY or ASCII. The “XX” part of the file name (FileXX) refers to the number of the instrument state. The first instrument state saved to any particular disk will be named “File00”, and each successive state saved to that disk will be numbered “1” higher than the previous state (for example, “File01” and “File02”).

Files with .i and .p File Extensions

The following two files (.i file and .p file) are always produced, except when **DATA ONLY**, is selected. These files were separated to allow backward compatibility with older instruments. The binary data contained in these two files is not meant to be read in an external computer:

- FileXX.i is a binary file, which contains the generic portion of the current instrument state (specifically, the System, Local, Preset, Copy, Save, and Sequence settings).
- FileXX.p is a binary file which contains portions of the instrument state specific to later instruments.

Files with .10, .11, .12, .1a, .1b, and .1c File Extensions

The following files are only produced if you have an active calibration. FileXX.10 is a binary file which stores the stimulus state of the instrument as it relates to an active calibration (specifically, the Power, Sweep Setup, Start, Stop, Center, and Span settings).

The same type of file is produced if Channel 2 is active, but the file extension is .20 instead of the .10 file extension mentioned in the previous paragraph.

Files FileXX.11 through .12, .1a, .1b, and .1c are binary files which contain the 12 error correction coefficients for Channel 1. If Channel 2 is active, it will have the same array, but file extensions are in the form .21, .22, .2a., .2b, and .2c. If you save in ASCII format, only .10 and .1c are produced, with .1c containing the entire error correction array in a two-column, real/imaginary (CITIfile) format.

Files with File Extensions .r1 through .r8

FileXX.r1 through .r4 are produced when **RAW ARRAY on OFF**, is turned ON. They may be either binary or ASCII, and contain the four raw uncorrected S-parameters for channel 1. Channel 2 has the same array, but file extensions are .r5 through .r8. In ASCII format, the data is displayed as two columns of real/imaginary numbers (CITIfile format); S11 appears first, S21 appears second, S12 appears third, and S22 appears last.

Files with .d1 and .d2 File Extensions

There are two type of files with .d1 and .d2 file extensions. There is FileXX.d1 (or .d2) and DataXX.d1 (or .d2).

FileXX.d1, produced only when **DATA ARRAY on OFF**, is turned ON, may be either binary or ASCII. This file contains the error-corrected measurement data, but without port extensions or electrical delay. In ASCII format, this is a two-column real/imaginary array (CITIfile

format), without any direct frequency information. S11 appears first, S21 second, S12 third, and S22 last. If Channel 2 is active, the same type of file is produced but the file extension is .d2. If dual display is on, both d1 and .d2 files are produced.

DataXX.d1, created only when **DATA ONLY on OFF** is turned ON, is either a binary file or an ASCII file in CITIfile format. Turning the **DATA ONLY on OFF** softkey ON suppresses the generation of all the previous file types. The contents of this file are identical to those of the FileXX.d1 file. The same type of file is produced if Channel 2 is active, but the file extension is .d2. If dual display is on, both .d1 and .d2 files are produced.

NOTE The DataXX files are much smaller than an entire instrument state, and are the best way to get just the data you want without saving the entire instrument state. Selecting more than one disk-save option does not confuse the analyzer, and simply produces all files associated with the selected options. The only exception to this is that selecting **DATA ONLY on OFF**, suppresses all other selected options.

Files with .s1 and .s2 File Extensions

There are two type of files with .s1 and .s2 file extensions. There is FileXX.s1 (or .s2) and DataXX.s1 (or .s2).

FileXX.s1 is an ASCII file in Touchstone S2P format. Basically, this is a file in real/imaginary spreadsheet type format with five columns: frequency in the first column, S11 in the second column, S21 in the third, S12 in the fourth, and S22 in the fifth column. If Channel 2 is active, the same type of file is produced but the file extension is .s2. If dual display is on, both .s1 and .s2 are produced. These Touchstone S2P files are only produced when a full 2-port calibration is active and **SAVE USING ASCII**, is selected. The effects of port extensions and electrical delay, if they are turned on, are included in the data.

DataXX.s1 is also an ASCII file in Touchstone S2P format. As with FileXX.s1, DataXX.s1 is a five-column real/imaginary spreadsheet type format where the columns are used the same as FileXX.s1. If Channel 2 is active, the same type of file is produced but the file extension is .s2. If dual display is on, both .s1 and .s2 are produced. These Touchstone S2P files are only produced when a full 2-port calibration is active and **SAVE USING ASCII**, is selected. The effects of port extensions and electrical delay, if they are turned on, are included in the data.

Files with .f1 and .f2 File Extensions

FileXX.f1, produced only when **FORMAT ARY on OFF**, is turned ON, may be either binary or ASCII. This file contains the formatted data in whichever format is currently displayed on the analyzer (dB, phase, VSWR, and so forth) with error correction, trace math, port extensions, electrical delay, time domain gating and smoothing applied. Port extensions are really only evident if the measured parameter is phase. The same type of file is produced for Channel 2, but the file extension is .f2. If dual display is on, both .f1 and .f2 are produced. In ASCII format, the data appears as two columns (CITIfile format). If the currently selected display format is not complex data (neither Smith Chart nor Polar), the second column will be meaningless (place holder) values.

Files with .g0 File Extension

FileXX.g0, produced only when **GRAPHICS on OFF**, is turned ON, is a binary file containing the active measurement trace and display graticule. The contents of this file are not meant to be read in an external computer, so this file is only of use in the instrument.

Binary Files

The size of the data files is very small, about one tenth the size compared to the ASCII format. Binary is the format to use when you want to store and recall instrument states on the analyzer quickly, but do not need to read the data in an external computer.

Viewing Files Within the Analyzer

All these files are rolled up into a single instrument state, so the analyzer shows only the "FileXX" part of the name, without an extension. The file description will say ISTATE, followed by parentheses with letters in them, such as (CDG). These letters are explained on the bottom of the analyzer screen, and indicate some of what is included in that instrument state. In this example, the state includes the Calibration Array, Data, and Graphics.

The only way to see all the file extensions previously described is to save the instrument state to a disk and view the file structure on an external computer.

Differences between Raw, Data, and Format Arrays

The following discussion explains the data processing flow in the analyzer. This information is very important if you will be utilizing data from your analyzer for use in computer applications such as spreadsheets, word processing, programs, etc. Refer to [Figure 6-1 on page 6-7](#).

The analyzer receives data from its A, B, and R or Aux-in inputs. Notice the three highlighted blocks. They represent the raw, data, and format arrays. You can save your measurement data in any or all of these format arrays each time the data is saved. Select the arrays of interest based on the factors discussed in this section.

For this discussion, only DISK saves will be described. Data can be saved to internal non-volatile memory or transferred over GPIB as well as to a floppy disk. You will find multiple files saved depending on the arrays chosen under the analyzer's **DEFINE DISK-SAVE**, menu. When using these files, it is important to know which file extension is needed for your particular job.

Raw Arrays

On the analyzer, press the **Save/Recall, DEFINE DISK-SAVE, RAW ARRAY ON**.

Data created the first time in this manner will be saved as filename "FILE00.r1". The file extension .r1 indicates the data was created while channel 1 was active and stored in the analyzer's raw data array. If you save the data again, but while channel 2 is active, you will get a new file called "FILE01.r2". RAW data are not commonly used unless sophisticated data processing is to be performed in an external PC. As an example, multi-port calibration is created by exporting raw data to a PC where error-correction for each of the multi-port paths is applied to them.

Data Arrays

Press **Save/Recall, DEFINE DISK-SAVE, DATA ARRAY ON**.

Data created the first time in this manner will be saved as filename "DATA00.d1." The file extension .d1 indicates that the data from the analyzer's channel 1 is error corrected data only if the analyzer's error correction feature is enabled (in other words, you have performed a calibration). Otherwise, the data is the same as data stored in the analyzer's raw data arrays. Data stored in the data arrays does not have any formatting applied to it.

Format Arrays

Press **Save/Recall, DEFINE DISK-SAVE, FORMAT ARY ON**,

Data created the first time in this manner will be saved as filename "FILE00.f1". The file extension .f1 indicates the data is formatted per [Figure 6-1 on page 6-7](#) using the analyzer's channel 1. Depending on what features you've selected, data in the format arrays includes data in the data arrays plus one or more of the following features:

- Trace math (i.e data-memory)
- Electrical delay
- Conversion (for complex impedance (Z), admittance (Y), etc.)
- Format (log, lin, phase, delay, SWR, excluding Smith and Polar)
- Smoothing

In each of these examples, most users will select **SAVE USING ASCII**, under the **DEFINE DISK-SAVE**, softkey menu. If **GRAPHICS on OFF**, is turned ON, an additional file will be created with file extension .g0. This is a Hewlett-Packard Graphics Language (HPGL) file.

Re-Saving an Instrument State

If you re-save a file, the analyzer overwrites the existing file contents.

NOTE You cannot re-save a file that contains data only. You must create a new file.

1. Press **Save/Recall**, **SELECT DISK**, and select the storage device.
 - INTERNAL MEMORY**
 - INTERNAL DISK**
 - EXTERNAL DISK** (If necessary, refer to the external disk setup procedure in [“Saving an Instrument State”](#) on page 6-5.)
2. Press **RETURN**, and then use the **up**, or **down**, key or the front-panel knob to highlight the name of the file that you want to re-save.
3. Press **RE-SAVE STATE, YES**.

Deleting a File

1. Press **Save/Recall, SELECT DISK**.
2. Choose from the following storage devices:
 - INTERNAL MEMORY**
 - INTERNAL DISK**
 - EXTERNAL DISK** (If necessary, refer to the external disk setup procedure in [“Saving an Instrument State” on page 6-5.](#))
3. Press **RETURN**.

To Delete an Instrument State File

- Press the **up**, or **down**, keys or the front panel knob to highlight the name of the file that you want to delete.
- Press **FILE UTILITIES, DELETE FILE, YES**, to delete all of the files that make up the selected instrument state.

To Delete all Files

- Press **FILE UTILITIES, DELETE ALL FILES, SAVE USING BINARY**, to delete all of the files that are on the selected storage device.

Renaming a File

1. Press **Save/Recall, AUTO-FEED OFF**.
2. Choose from the following storage devices:
 - INTERNAL MEMORY**
 - INTERNAL DISK**
 - EXTERNAL DISK** (If necessary, refer to the external disk setup procedure in [“Saving an Instrument State” on page 6-5.](#))
3. Press **RETURN**, and then use the **up**, or **down** keys or the front panel knob to highlight the name of the file that you want to rename.
4. Press **RETURN, FILE UTILITIES, RENAME FILE, ERASE TITLE**.
5. Turn the front panel knob to point to each character of the new file name, pressing **SELECT LETTER** when the arrow points to each character. Press **BACK SPACE**, if you enter an incorrect character. After you have selected all the characters in the new file name, press **DONE**.

NOTE Renaming files may also be done by using the optional external keyboard.

Recalling a File

1. Press **Save/Recall, SELECT DISK**.
2. Choose from the following storage devices:
 - INTERNAL MEMORY**
 - INTERNAL DISK**
 - EXTERNAL DISK** (If necessary, refer to the external disk setup procedure in [“Saving an Instrument State” on page 6-5.](#))
3. Press the **up**, or **down**, keys or the front panel knob to highlight the name of the file that you want to recall.
4. Press **RETURN, RECALL STATE**.

Formatting a Disk

1. Press **Save/Recall**, **FILE UTILITIES**, **FORMAT DISK**.
2. Choose the type of format you want:
 - FORMAT:LIF**
 - FORMAT:DOS**
3. Press **FORMAT DISK**, **YES**.

Solving Problems with Saving or Recalling Files

If you encounter a problem when you are storing files to disk, or the analyzer internal memory, check the following list for possible causes:

- Look in the analyzer display message area. The analyzer may show a message that will identify the problem. Refer to the "Error Messages" chapter of the reference guide if you view a message.
- Make sure that you are NOT using a single-sided floppy disk in the analyzer disk drive.
- Make sure that you are using a formatted disk.
- Make sure that the disk has not been formatted with the LIF-HFS (hierarchical file system) extensions as the analyzer does not support this format.

If You Are Using an External Disk Drive

- Make sure that the analyzer is in system controller mode, by pressing **Local, SYSTEM CONTROLLER**.
- Make sure that you have connected the disk drive to ac power, switched on the power, and connected a GPIB cable between the disk drive and the analyzer.
- Make sure that the analyzer recognizes the disk drive's GPIB address, as explained in [“If You Are Plotting Measurement Results to a Disk Drive” on page 7-13](#).
- Make sure that the analyzer recognizes the disk (drive) unit that you selected (0 or 1).
- If the external disk is a hard disk, make sure that the disk volume number is set correctly.
- If the disk drive is an older HP 9122, it may not recognize the newer high density disks.
- Substitute the GPIB cable.
- Substitute the disk drive.

7

- “Printing Measurements Results” on page 7-2
- “Configuring a Print Function” on page 7-4
- “Defining a Print Function” on page 7-6
- “Printing One Measurement Per Page” on page 7-8
- “Printing Multiple Measurements Per Page” on page 7-9
- “Configuring a Plot Function” on page 7-10
- “Defining a Plot Function” on page 7-15
- “Plotting One Measurement Per Page Using a Pen Plotter” on page 7-19
- “Plotting Multiple Measurements Per Page Using a Pen Plotter” on page 7-20
- “To View Plot Files on a PC” on page 7-22
- “Outputting Plot Files from a PC to a Plotter” on page 7-23
- “Outputting Plot Files from a PC to an HPGL Compatible Printer” on page 7-24
- “Outputting Single Page Plots Using a Printer” on page 7-26
- “Outputting Multiple Plots to a Single Page Using a Printer” on page 7-27
- “Plotting Multiple Measurements Per Page from Disk” on page 7-28
- “Titling the Displayed Measurement” on page 7-32
- “Configuring the Analyzer to Produce a Time Stamp” on page 7-33
- “Aborting a Print or Plot Process” on page 7-34
- “Printing or Plotting the List Values or Operating Parameters” on page 7-35
- “Solving Problems with Printing or Plotting” on page 7-36

Printing Measurement Results

Printing Measurements Results

In this chapter, you can find procedures for configuring the equipment, defining the format, and outputting your measurement results to a printer or plotter.

Printing or Plotting Your Measurement Results

You can print your measurement results to the following peripherals:

- printers with GPIB interfaces
- printers with parallel interfaces
- printers with serial interfaces

You can plot your measurement results to the following peripherals:

- HPGL compatible printers with GPIB interfaces
- HPGL compatible printers with parallel interfaces
- plotters with GPIB interfaces
- plotters with parallel interfaces
- plotters with serial interfaces

Most Hewlett-Packard desktop printers and plotters are compatible with the analyzer. For a list of recommended peripherals, refer to the configuration guide for your analyzer. The following Web site also contains a link to the configuration guide:

www.agilent.com/find/8703

A printer compatibility guide (an up-to-date list of printers that are compatible with the analyzer) can be found at the following Web site:

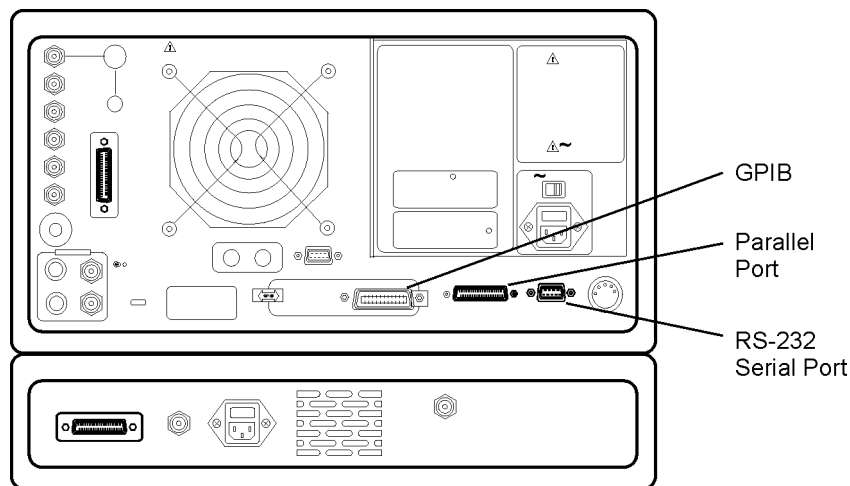
www.agilent.com/find/pcg

Configuring a Print Function

All copy configuration settings are stored in non-volatile memory. Therefore, they are not affected if you press **Preset**, or switch off the analyzer power.

1. Connect the printer to the interface port.

Figure 7-1. Printer Connections to the Analyzer



rpl_print

2. Press **Local**, **SET ADDRESSES**, **PRINTER PORT**, **PRNTR TYPE**, until the correct printer choice appears:
 - ThinkJet (QuietJet)
 - DeskJet (This supports most current models such as DeskJet 890C, DeskJet 895C, or DeskJet 1600C. See also DJ 540 selection.)
 - LaserJet (only LaserJet models III, 4, 5, and 6)
 - PaintJet
 - Epson-P2 (printers that conform to the ESC/P2 printer control language, such as Epson LQ-570)
 - DJ 540 (This can be used for printers that do not support 100 dots per inch (dpi) but do support 300 dpi such as HP DeskJet 540 or 850C.)

NOTE Selecting DJ 540 converts 100 dpi raster information to 300 dpi raster format. If your DeskJet printer does not support the 100 dpi raster format and your printing results seem to be smaller than the normal size (approximately one-half of the page) select DJ540.

3. Select one of the following printer interfaces:

- Choose **PRNTR PORT GPIB**, if your printer has a GPIB interface, and then configure the print function as follows:
 - a. Enter the GPIB address of the printer, followed by **x1**.
 - b. Press **Local**, and **SYSTEM CONTROLLER**, if there is no external controller connected to the GPIB bus.
 - c. Press **Local**, and **USE PASS CONTROL**, if there is an external controller connected to the GPIB bus.

- Choose **PARALLEL**, if your printer has a parallel (Centronics) interface, and then configure the print function as follows:

Press **Local**, and then select the parallel port interface function by pressing **PARALLEL**, until the correct function appears.

- If you choose **PARALLEL [COPY]**, the parallel port is dedicated for normal copy device use (printers or plotters).
 - If you choose **PARALLEL [GPIO]**, the parallel port is dedicated for general purpose I/O, and cannot be used for printing or plotting.
- Choose **SERIAL**, if your printer has a serial (RS-232) interface, and then configure the print function as follows:
 - a. Press **PRINTER BAUD RATE**, and enter the printer's baud rate, followed by **x1**.
 - b. To select the transmission control method that is compatible with your printer, press **XMIT CNTRL**, (transmit control - handshaking protocol) until the correct method appears.
 - If you choose **Xon-Xoff**, the handshake method allows the printer to control the data exchange by transmitting control characters to the analyzer.
 - If you choose **DTR-DSR**, the handshake method allows the printer to control the data exchange by setting the electrical voltage on one line of the RS-232 serial cable.

NOTE Because the **DTR-DSR**, handshake takes place in the hardware rather than the firmware or software, it is the fastest transmission control method.

Defining a Print Function

NOTE The print definition is set to default values whenever the power is cycled. However, you can save the print definition by saving the instrument state.

1. Press **Copy**, **DEFINE PRINT**.
 2. Press **PRINT: MONOCHROME**, or **PRINT: COLOR**.
 - Choose **PRINT: MONOCHROME**, if you are using a black and white printer, or you want just black and white from a color printer.
 - Choose **PRINT: COLOR**, if you are using a color printer.
 3. Press **AUTO-FEED**, until the correct choice (ON or OFF) is highlighted.
 - Choose **AUTO-FEED ON**, if you want to print one measurement per page.
 - Choose **AUTO-FEED OFF**, if you want to print multiple measurements per page.
-

NOTE Laser printers and some DeskJet printers do not begin to print until a full page, or a partial page and a form feed, have been received.

If You Are Using a Color Printer

1. Press **PRINT COLORS**.
 2. If you want to modify the print colors, select the print element and then choose an available color.
-

NOTE Since the media color is usually white or clear, you could set a print element to white if you do not want that element to appear on your hardcopy.

To Reset the Printing Parameters to Default Values

1. Press **Copy**, **DEFINE PRINT**, **DEFAULT PRNT SETUP**.

Table 7-1. Default Values for Printing Parameters

Printing Parameter	Default
Printer Mode	Monochrome
Auto Feed	ON
Printer Colors	
Channel 1 and 3 Data	Magenta
Channel 1 and 3 Memory	Green
Channel 2 and 4 Data	Blue
Channel 2 and 4 Memory	Red
Graticule	Cyan
Warning	Black
Text	Black
Ref Line	Black

Printing One Measurement Per Page

1. Configure and define the print function, as explained in “[Configuring a Print Function](#)” on page 7-4 and “[Defining a Print Function](#)” on page 7-6.
2. Press **Copy**, **PRINT MONOCHROME**.

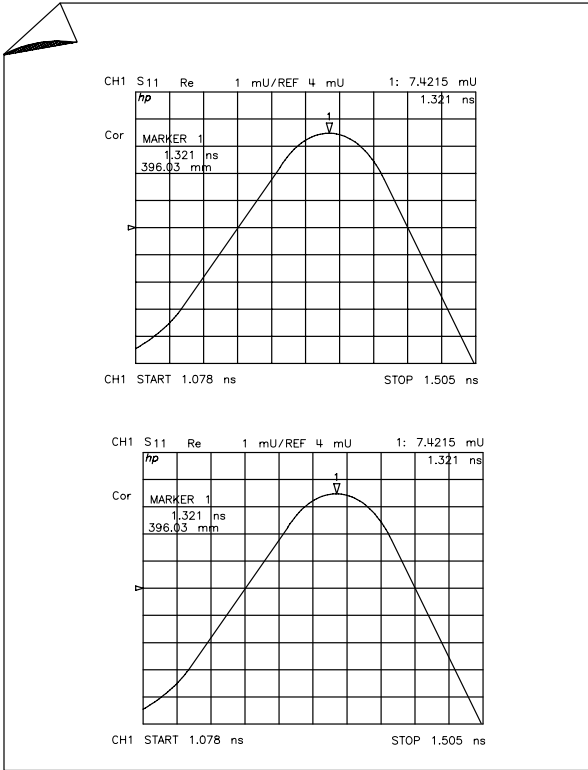
If you defined the **AUTO-FEED OFF**, press **PRINTER FORM FEED**, after the message COPY OUTPUT COMPLETED appears.

Printing Multiple Measurements Per Page

1. Configure and define the print function, as explained in “Configuring a Print Function” on page 7-4 and “Defining a Print Function” on page 7-6.
2. Press **Copy**, **DEFINE PRINT**, and then press **AUTO-FEED**, until the softkey label appears as **AUTO-FEED OFF**.
3. Press **RETURN**, **PRINT MONOCHROME**, to print a measurement on the first half page.
4. Make the next measurement that you want to see on your hardcopy. [Figure 7-2](#) shows an example of a hardcopy where two measurements appear.
5. Press **Copy**, **PRINT MONOCHROME**, to print a measurement on the second half page.

NOTE This feature will not work for all printers due to differences in printer resolution.

Figure 7-2. Printing Two Measurements



pg6148d

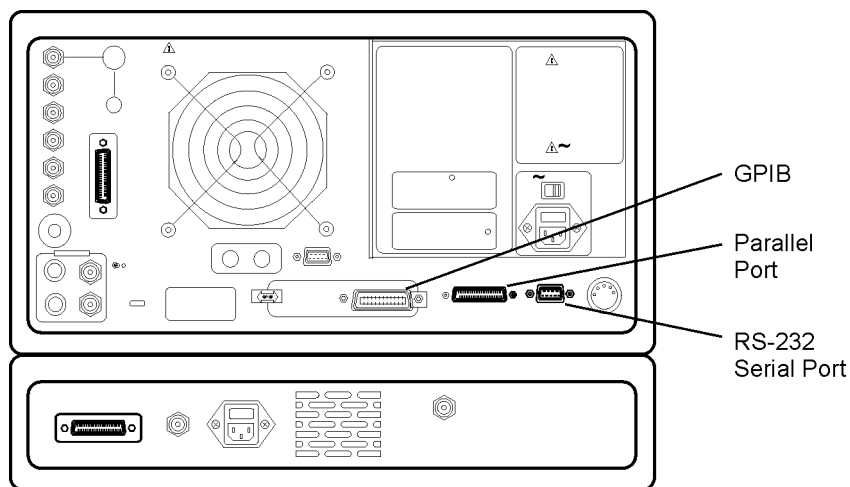
Configuring a Plot Function

All copy configuration settings are stored in non-volatile memory. Therefore, they are not affected if you press **Preset**, or switch off the analyzer power.

Peripheral Interface	Recommended Cables
Parallel	92284A
GPIB	10833A/33B/33D
Serial	24542G

1. Connect the peripheral to the interface port using the recommended cable from the following list.

Figure 7-3. Peripheral Connections to the Analyzer



rpl_print

If You Are Plotting to an HPGL/2 Compatible Printer

2. Press **Local**, **SET ADDRESSES**, **PLTR PORT**, and then press **PLTR TYPE []** until **HPGL PRT** appears.

3. Configure the analyzer for one of the following printer interfaces:

- Choose **PRNTR PORT GPIB** if your printer has an GPIB interface, and then configure the print function as follows:
 - a. Enter the GPIB address of the printer (default is 01), followed by **x1**.
 - b. Press **Local**, and **SYSTEM CONTROLLER** if there is no external controller connected to the GPIB bus.
 - c. Press **Local**, and **USE PASS CONTROL** if there is an external controller connected to the GPIB bus.
- Choose **PARALLEL** if your printer has a parallel (Centronics) interface, and then configure the print function as follows:

Press **Local**, and then select the parallel port interface function by pressing **PARALLEL** until the correct function appears.

 - If you choose **PARALLEL [COPY]**, the parallel port is dedicated for normal copy device use (printers or plotters).
 - If you choose **PARALLEL [GPIO]**, the parallel port is dedicated for general purpose I/O, and cannot be used for printing or plotting.
- Choose **SERIAL**, if your printer has a serial (RS-232) interface, and then configure the print function as follows:
 - a. Press **PRINTER BAUD RATE**, and enter the printer's baud rate, followed by **x1**.
 - b. To select the transmission control method that is compatible with your printer, press **XMIT CNTRL**, (transmit control - handshaking protocol) until the correct method appears.
 - If you choose **Xon-Xoff**, the handshake method allows the printer to control the data exchange by transmitting control characters to the analyzer.
 - If you choose **DTR-DSR**, the handshake method allows the printer to control the data exchange by setting the electrical voltage on one line of the RS-232 serial cable.

NOTE Because the **DTR-DSR**, handshake takes place in the hardware rather than the firmware or software, it is the fastest transmission control method.

4. Press **Local**, **SET ADDRESSES**, **PLOTTER PORT**, and then **PLTR TYPE**, until **PLTR TYPE [HPGL PRT]**, appears.

If You Are Plotting to a Pen Plotter

1. Press **Local**, **SET ADDRESSES**, **PLOTTER PORT**, and then **PLTR TYPE**, until **PLTR TYPE [PLOTTER]**, appears.

2. Configure the analyzer for one of the following plotter interfaces:

- Choose **PLTR PORT GPIB**, if your plotter has a GPIB interface, and then configure the plot function as follows:
 - a. Enter the GPIB address of the plotter (default is 05), followed by **x1**.
 - b. Press **Local**, and **SYSTEM CONTROLLER**, if there is no external controller connected to the GPIB bus.
 - c. Press **Local**, and **USE PASS CONTROL**, if there is an external controller connected to the GPIB bus.
- Choose **PARALLEL**, if your plotter has a parallel (Centronics) interface, and then configure the plot function as follows:
 - Press **Local**, and then select the parallel port interface function by pressing **PARALLEL**, until the correct function appears.
 - If you choose **PARALLEL [COPY]**, the parallel port is dedicated for normal copy device use (printers or plotters).
 - If you choose **PARALLEL [GPIO]**, the parallel port is dedicated for general purpose I/O, and cannot be used for printing or plotting.
- Choose **SERIAL**, if your plotter has a serial (RS-232) interface, and then configure the plot function as follows:
 - a. Press **PLOTTER BAUD RATE**, and enter the plotter's baud rate, followed by **x1**.
 - b. To select the transmission control method that is compatible with your plotter, press **XMIT CNTRL**, (transmit control - handshaking protocol) until the correct method appears.
 - If you choose **Xon-Xoff**, the handshake method allows the plotter to control the data exchange by transmitting control characters to the analyzer.
 - If you choose **DTR-DSR**, the handshake method allows the plotter to control the data exchange by setting the electrical voltage on one line of the RS-232 serial cable.

NOTE Because the **DTR-DSR**, handshake takes place in the hardware rather than the firmware or software, it is the fastest transmission control method.

If You Are Plotting Measurement Results to a Disk Drive

The plot files that you generate from the analyzer, contain the HPGL representation of the measurement display. The files will not contain any setup or formfeed commands.

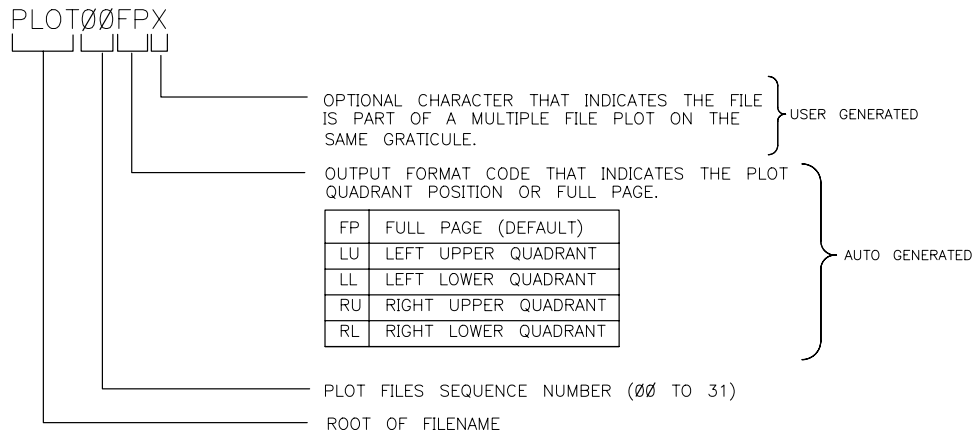
CAUTION Do not mistake the line switch for the disk eject button when you are removing the disk from the analyzer. If the line switch is mistakenly pushed, the instrument will be turned off, losing all settings and data that have not been saved.

1. Configure the analyzer to plot to disk.
 - a. Press **Local, SET ADDRESSES, PLOTTER PORT, DISK.**
 - b. Press **Save/Recall, SELECT DISK,** and select the disk drive that you will plot to.
 - Choose **DEFINE PRINT,** if you will plot to the analyzer internal disk drive.
 - Choose **EXTERNAL DISK,** if you will plot to a disk drive that is external to the analyzer. Then configure the disk drive as follows:
 1. Press **CONFIGURE EXT DISK, ADDRESS: DISK,** and enter the GPIB address to the disk drive (default is 00) followed by **x1.**
 2. Press **Local, DISK UNIT NUMBER,** and enter the drive where your disk is located, followed by **x1.**
 3. If your storage disk is partitioned, press **VOLUME NUMBER,** and enter the volume number where you want to store the instrument state file.
2. Press **Copy, PLOT.**

The analyzer assigns the first available default filename for the displayed directory. For example, the analyzer would assign PLOT00FP for a LIF format (PLOT00.FP for a DOS format) if there were no previous plot files saved to the disk.

Figure 7-4 shows the three parts of the file name that are generated automatically by the analyzer whenever a plot is requested. The two digit sequence number is incremented by one each time a file with a default name is added to the directory.

Figure 7-4. Automatic File Naming Convention for LIF Format



ph646c

To Output the Plot Files

- You can plot the files to a plotter from a personal computer.
- You can output your plot files to an HPGL compatible printer, by following the sequence in [“Outputting Plot Files from a PC to an HPGL Compatible Printer”](#) on page 7-24.
- You can run a program that plots all of the files, with the root filename of `PLOT`, to an HPGL compatible printer. This program is provided on the CD-ROM of example programs that is included in the programmer’s guide. However, this program is for use with LIF formatted disks and is written in HP BASIC.

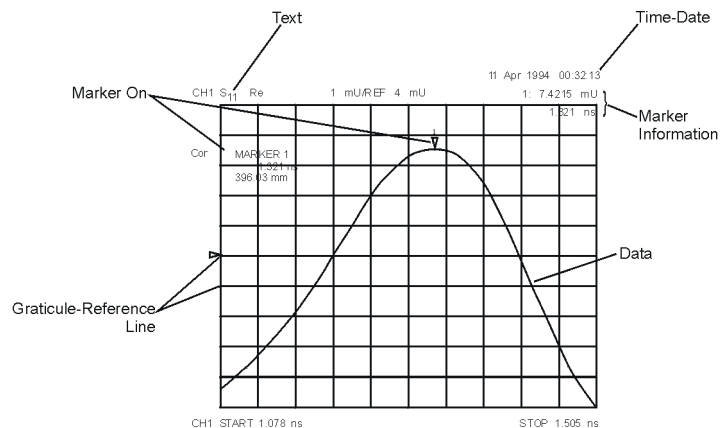
Defining a Plot Function

1. Press **Copy**, **DEFINE PLOT**.

Choosing Display Elements

- Choose which of the following measurement display elements that you want to appear on your plot:
 - Choose **PLOT DATA ON**, if you want the measurement data trace to appear on your plot.
 - Choose **PLOT MEM ON**, if you want the displayed memory trace to appear on your plot.
 - Choose **PLOT GRAT ON**, if you want the graticule and the reference line to appear on your plot.
 - Choose **PLOT TEXT ON**, if you want all of the displayed text to appear on your plot. (This does not include the marker values or softkey labels.)
 - Choose **PLOT MKR ON**, if you want the displayed markers, and marker values, to appear on your plot.

Figure 7-5. Plot Components Available through Definition



pa5167e

Selecting Auto-Feed

- Press **AUTO-FEED**, until the correct choice is highlighted.
 - Choose **AUTO-FEED ON**, if you want a “page eject” sent to the plotter or HPGL compatible printer after each time you press **PLOT**.
 - Choose **AUTO-FEED OFF**, if you want multiple plots on the same sheet of paper.

NOTE The peripheral ignores **AUTO-FEED ON**, when you are plotting to a quadrant.

Selecting Pen Numbers and Colors

- Press **MORE**, and select the plot element where you want to change the pen number. For example, press **PEN NUM DATA**, and then modify the pen number. The pen number selects the color if you are plotting to an HPGL/2 compatible color printer.

Press **x1**, after each modification.

NOTE The following color assignments are valid for HPGL/2 compatible color printers only. When using word processor or graphics presentation programs, different colors may be assigned to the pen numbers.

Table 7-2. Default Pen Numbers and Corresponding Colors

Pen Number	Color	Pen Number	Color
0	white	4	yellow
1	cyan	5	green
2	magenta	6	red
3	blue	7	black

Table 7-3. Default Pen Numbers for Plot Elements

Corresponding Key	Plot Element	Pen Numbers	
		Channel 1	Channel 2
PEN NUM DATA,	Measurement Data Trace	2	3
PEN NUM MEMORY,	Displayed Memory Trace	5	6
PEN NUM GRATICULE,	Graticule and Reference Line	1	1
PEN NUM TEXT,	Displayed Text	7	7
PEN NUM MARKER,	Displayed Markers and Values	7	7

NOTE You can set all the pen numbers to black for a plot in black and white.
You must define the pen numbers separately for each measurement channel (channel 1/3 and channel 2/4).

Selecting Line Types

- Press **MORE**, and select each plot element line type that you want to modify.
 - Select **LINE TYPE DATA**, to modify the line type for the data trace. Then enter the new line type (see [Figure 7-6](#)), followed by **x1**.
 - Select **LINE TYPE MEMORY**, to modify the line type for the memory trace. Then enter the new line type (see [Figure 7-6](#)), followed by **x1**.

Table 7-4. Default Line Types for Plot Elements

Plot Elements	Channel 1 Line Type Numbers	Channel 2 Line Type Numbers
Data Trace	7	7
Memory Trace	7	7

Figure 7-6. Line Types Available

- 0- Specifies dots only at the points that are plotted.
- 1-
- 2- — — — — —
- 3- — — — — —
- 4- — — — — —
- 5- — — — — —
- 6- — — — — —
- 7- — — — — —

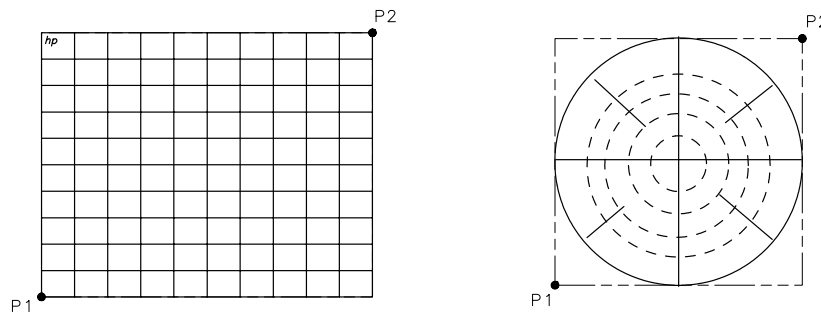
pg6135d

NOTE You must define the line types for each measurement channel (channel 1/3 and channel 2/4).

Choosing Scale

- Press **SCALE PLOT**, until the selection appears that you want.
 - Choose **SCALE PLOT [FULL]**, if you want the normal scale selection for plotting. This includes space for all display annotations such as marker values and stimulus values. The entire analyzer display fits within the defined boundaries of P1 and P2 on the plotter, while maintaining the exact same aspect ratio as the display.
 - Choose **SCALE PLOT [GRAT]**, if you want the outer limits of the graticule to correspond to the defined P1 and P2 scaling point on the plotter. (Intended for plotting on preprinted rectangular or polar forms.)

Figure 7-7. Locations of P1 and P2 in SCALE PLOT [GRAT], Mode



pg6157d

Choosing Plot Speed

- Press **PLOT SPEED**, until the plot speed appears that you want.
 - Choose **PLOT SPEED [FAST]** for normal plotting.
 - Choose **PLOT SPEED [SLOW]** for plotting directly on transparencies. (The slower speed provides a more consistent line width.)

To Reset the Plotting Parameters to Default Values

Press **Copy, DEFINE PLOT, MORE, MORE, YES.**

Table 7-5. Plotting Parameter Default Values

Plotting Parameter	Default Value	Plotting Parameter	Default Value
Select Quadrant	Full page	Plot Scale	Full
Auto Feed	ON	Plot Speed	Fast
Define Plot	All plot elements on	Line Type	7 (solid line)
Pen Numbers: Channel 1 and 3		Pen Numbers: Channel 2 and 4	
Data	2	Data	3
Memory	5	Memory	6
Graticule	1	Graticule	1
Text	7	Text	7
Marker	7	Marker	7

Plotting One Measurement Per Page Using a Pen Plotter

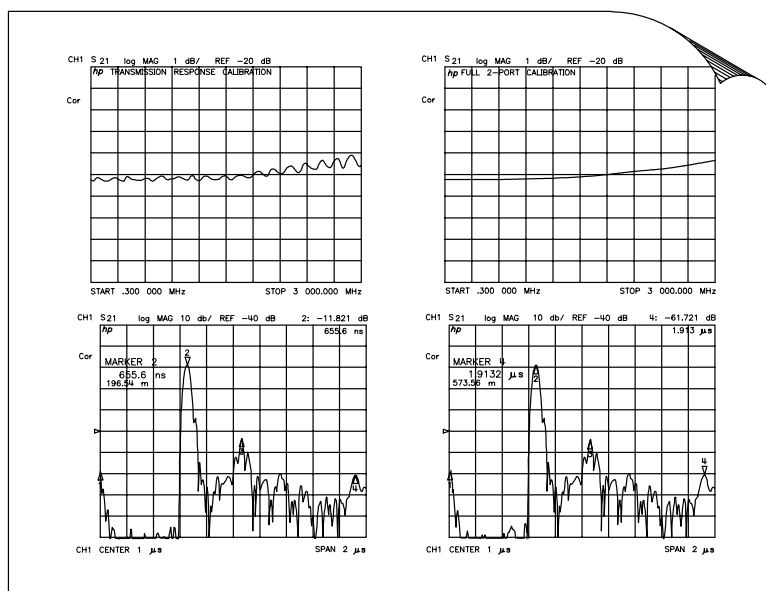
1. Configure and define the plot, as explained in “[Configuring a Plot Function](#)” on page 7-10 and “[Defining a Plot Function](#)” on page 7-15.
2. Press **Copy, PLOT**.
 - If you defined the **AUTO-FEED OFF**, press **PLOTTER FORM FEED**, after the message **COPY OUTPUT COMPLETED** appears.

Plotting Multiple Measurements Per Page Using a Pen Plotter

1. Configure and define the plot, as explained in “Configuring a Plot Function” on page 7-10 and “Defining a Plot Function” on page 7-15.
2. Press **Copy**, **SEL QUAD []**.
3. Choose the quadrant where you want your displayed measurement to appear on the hardcopy. The following quadrants are available:
 - LEFT UPPER**
 - LEFT LOWER**
 - RIGHT UPPER**
 - RIGHT LOWER**

The selected quadrant will appear in the brackets under **SEL QUAD []**.

Figure 7-8. Plot Quadrants



pg65e

4. Press **PLOT**.
5. Make the next measurement that you want to see on your hardcopy.
6. Press **Copy**, and choose another quadrant where you want to place the displayed measurement.
7. Repeat the previous three steps until you have captured the results of up to four measurements.

If You Are Plotting to an HPGL Compatible Printer

1. Configure and define the plot, as explained in “Configuring a Plot Function” on page 7-10 and “Defining a Plot Function” on page 7-15.
2. Press **Copy, PLOT, PLOTTER FORM FEED**, to print the data the printer has received.

NOTE Use test sequencing to automatically plot all four S-parameters.

1. Set all measurement parameters.
2. Perform a full 2-port calibration.
3. Enter the test sequence:

Seq, NEW SEQ/MODIFY SEQ, SEQUENCE 1 SEQ1

Meas, Refl: FWD S11 (A/R)

Copy, SEL QUAD [], SELECT DISK, LEFT UPPER PLOT

Meas, Trans: FWD S21 (B/R)

Copy, SEL QUAD [], LEFT LOWER PLOT

Meas, Refl: REV S22 (B/R)

Copy, SEL QUAD [], RIGHT UPPER PLOT

Meas, Trans: REV S12 (B/R)

Copy, SEL QUAD [], RIGHT LOWER PLOT

Seq, DONE SEQ MODIFY

4. Run the test sequence by pressing:

Seq, DO SEQUENCE, SEQUENCE 1 SEQ1

To View Plot Files on a PC

Plot files can be viewed and manipulated on a PC using a word processor or graphics presentation program. Plot files contain a text stream of HPGL (Hewlett-Packard Graphics Language) commands. In order to import a plot file into an application, that application *must* have an import filter for HPGL (often called HGL).

When viewed in PC programs, the color and font size of the plot may vary from the output of an HPGL/2 compatible color printer. Refer to [“Selecting Pen Numbers and Colors” on page 7-16](#)

To modify the color or font size, consult the documentation for the particular application being used.

NOTE Plot files may also be saved to a floppy disk as a JPEG file and used on a personal computer. Refer to [“Saving in Graphical \(JPEG\) Form” on page 6-13](#).

Converting HPGL Files for Use with Other PC Applications

A utility can convert hpgl (or .fp) files to other PC applications. This utility, named hp2xx, is available to be downloaded without charge (on donation basis only) from Free Software Foundation. You may download this file using the information available on the following Web site: <ftp://ots.external.hp.com/rfmw/lou/>

To convert HPGL files to be used with other PC applications.

1. Using the instructions on the Free Software Foundation website, FTP the hp2xx file and save it on a floppy disk as “hp2xx.exe”.
2. Create the following batch file and save it on the same floppy disk as “hpglconv.bat”.

The batch file consists of the following two lines:

```
@ echo off  
A:\hp2xx.exe -m pcx %1
```

(where “A” is the disk drive where the floppy disk is installed.)

3. Insert the floppy disk (with the two files already installed) into your analyzer.
4. Make sure the measurement that you want to convert is displayed on the analyzer display.
5. Create an HPGL file of the measurement and save it to the floppy disk by pressing **Local, SET ADDRESSES, PLOTTER PORT, DISK, Copy, PLOT**.
6. Remove the floppy disk from the analyzer and insert it back into the PC.
7. Using Explorer or File Manager, click and drag the icon of the newly-created hpgl file onto the icon of the hpglconv.bat file. This process creates a PCX format file from the hpgl file.

NOTE This conversion method has been used to convert many measurement displays. However, this conversion utility is not supported by Agilent Technologies.

Outputting Plot Files from a PC to a Plotter

1. Connect the plotter to an output port of the computer (for example, COM1).
2. If using the COM1 port, output the file to the plotter by using the following command:

```
C:> TYPE PLOT00.FP > COM1
```

Outputting Plot Files from a PC to an HPGL Compatible Printer

To output the plot files to an HPGL compatible printer, you can use the HPGL initialization sequence linked in a series as follows:

- Step 1. Store the HPGL initialization sequence in a file named `hpglinit`.
- Step 2. Store the exit HPGL mode and form feed sequence in a file named `exithpgl`.
- Step 3. Send the HPGL initialization sequence to the printer.
- Step 4. Send the plot file to the printer.
- Step 5. Send the exit HPGL mode and form feed sequence to the printer.

Step 1. Store the HPGL initialization sequence.

1. Create a test file by typing in each character as shown in the left column of [Table 7-6](#). Do not insert spaces or linefeeds. Most editors allow the inclusion of escape sequences.

For example, in the MS-DOS editor (DOS 5.0 or greater), press CNTRL-P (hold down the CTRL key and press P) followed by the ESCape key to create the escape character.

2. Name the file `hpglinit`.

Table 7-6. HPGL Initialization Commands

Command	Remark
<code><esc>E</code>	conditional page eject
<code><esc>&12A</code>	page size 8.5 x 11
<code><esc>&l1O</code>	landscape orientation (lower case l, one, capital O)
<code><esc>&a0L</code>	no left margin (a, zero, capital L)
<code><esc>&a400M</code>	no right margin (a, 4, zero, zero, capital M)
<code><esc>&l0E</code>	no top margin (lower case l, zero, capital E)
<code><esc>*c7680x5650Y</code>	frame size 10.66"x 7.847" (720 decipoints/inch)
<code><esc>*p50x50Y</code>	move cursor to anchor point
<code><esc>*c0T</code>	set picture frame anchor point
<code><esc>*r-3U</code>	set CMY palette
<code><esc>%1B</code>	enter HPGL mode; cursor at PCL

NOTE As shown in [Table 7-6](#), the `<esc>` is the symbol used for the escape character, decimal value 27.

Step 2. Store the exit HPGL mode and form feed sequence.

1. Create a test file by typing in each character as shown in the left column of [Table 7-7](#). Do not insert spaces or linefeeds.
2. Name the file `exithpgl`.

Table 7-7. HPGL Test File Commands

Command	Remark
<code><esc>%0A</code>	exit HPGL mode
<code><esc>E</code>	form feed

Step 3. Send the HPGL initialization sequence to the printer.

Type `print hpglinit` to send the initialization sequence to the printer.

Step 4. Send the plot file to the printer.

Type `print filename` (where `filename` is the name of the HPGL plot file) to send the plot file to the printer.

Step 5. Send the exit HPGL mode and form feed sequence to the printer.

Type `print exithpgl` to send the HPGL mode and form feed sequence to the printer.

Outputting Single Page Plots Using a Printer

You can output plot files to an HPGL compatible printer using the DOS command line and the files created in the previous steps. This example assumes that the escape sequence files and the plot files are in the current directory and the selected printer port is PRN.

Command	Remarks
C:>	type hpglinit > PRN
C:>	type PLOT00.FP> PRN
C:>	type exithpgl > PRN

Outputting Multiple Plots to a Single Page Using a Printer

Refer to “[Plotting Multiple Measurements Per Page from Disk](#)” on page 7-28 for the naming conventions for plot files that you want printed on the same page. You can use the following batch file to automate the plot file printing. In this example, the batch file is saved as “do_plot.bat.” However, before running this batch file, you must first create the hpglinit file and the exithpgl file described in “[Outputting Plot Files from a PC to an HPGL Compatible Printer](#)” on page 7-24.

```
rem _____  
rem Name: do_plot  
rem  
rem Description:  
rem  
rem output HPGL initialization sequence to a file:spooler  
rem append all the requested plot files to the spooler  
rem append the formfeed sequence to the spooler  
rem copy the file to the printer  
rem  
rem (This routine uses COPY instead of PRINT because COPY  
rem will not return until the action is completed. PRINT  
rem will queue the file so the subsequent DEL will likely  
rem generate an error. COPY avoids this.)  
rem _____
```

```
echo off  
type hpglinit > spooler  
for %%i in (%1) do type %%i >> spooler  
type exithpgl >> spooler  
copy spooler LPT1  
del spooler  
echo on
```

For example, you have the following list of files to plot:

```
PLOT00.LL  
PLOT00.LU  
PLOT00.RL  
PLOT00.RU
```

You would invoke the batch print as follows:

```
C:> do_plot PLOT00.*
```

Plotting Multiple Measurements Per Page from Disk

The following procedures show you how to store plot files on a LIF formatted disk. A naming convention is used so you can later run an HP BASIC program on an external controller that will output the files to the following peripherals:

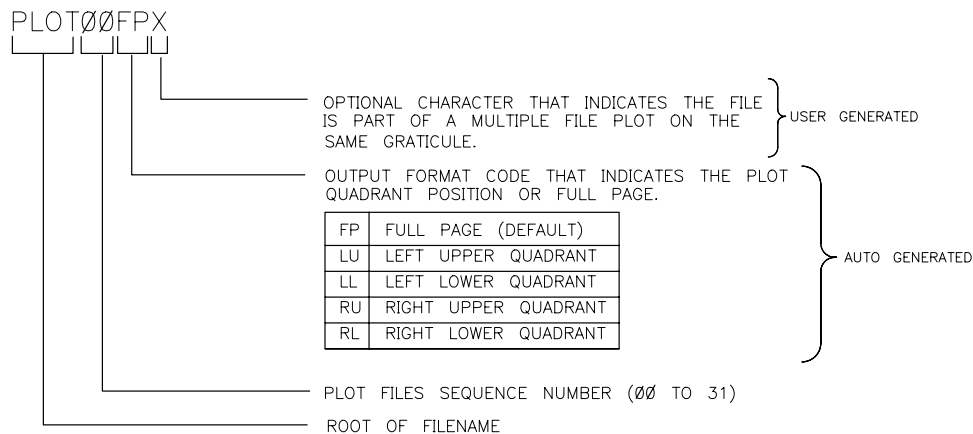
- a plotter with auto-feed capability, such as the HP 7550B
- an HP-GL/2 compatible printer, such as the LaserJet 4 series (monochrome) or the DeskJet 1200C or DeskJet 1600C (color)

The program is provided on the CD-ROM that includes example programs and the analyzer documentation. The file naming convention allows the program to initiate the following:

- to initialize the printer for HP-GL/2 at the beginning of a page
- to plot multiple plot files on the same page
- to send a page eject (form feed) to the hardcopy device, when all plots to the same page have been completed

The plot file name is made up of four parts; the first three are generated automatically by the analyzer whenever a plot is requested. The two digit sequence number is incremented by one each time a file with a default name is added to the directory.

Figure 0-1 Plot Filename Convention



ph646c

To Plot Multiple Measurements on a Full Page

You may want to plot various files to the same page, for example, to show measurement data traces for different input settings, or parameters, on the same graticule.

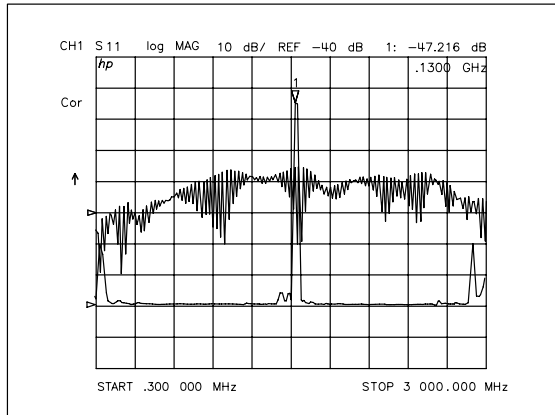
1. Define the plot, as explained in [“Defining a Plot Function” on page 7-15](#).
2. Press **Copy, PLOT, DEFINE PRINT**. The analyzer assigns the first available default filename for the displayed directory. For example, the analyzer would assign PLOT00FP if there were no previous plot files on the disk.
3. Press **Save/Recall**, and turn the front panel knob to highlight the name of the file that you just saved.
4. Press **FILE UTILITIES, RENAME FILE**, and turn the front panel knob to place the ↑ pointer to the A character.
5. Press **SELECT LETTER, DONE**.
6. Define the next measurement plot that you will be saving to disk.

For example, you may want only the data trace to appear on the second plot for measurement comparison. In this case, you would press **Copy, DEFINE PLOT**, and choose **PLOT DATA ON, PLOT MEM OFF, PLOT GRAT OFF, PLOT TEXT OFF, PLOT MKR OFF**.
7. Press **Copy, PLOT**. The analyzer will assign PLOT00FP because you renamed the last file saved.
8. Press **Save/Recall**, and turn the front panel knob to highlight the name of the file that you just saved.
9. Press **FILE UTILITIES, RENAME FILE**, and turn the front panel knob to place the ↑ pointer to the B character.
10. Press **SELECT LETTER, DONE**.
11. Continue defining plots and renaming the saved file until you have saved all the data that you want to put on the same page. Renaming the files as shown allows you to use the provided program, that organizes and plots the files, according to the file naming convention.

Plot File	Recognized Filename
First File Saved	PLOT00FPA
Second File Saved	PLOT00FPB
Third File Saved	PLOT00FPC
Fourth File Saved	PLOT00FPD

Figure 7-9 shows plots for two responses of the same device.

Figure 7-9. Plotting Two Files on the Same Page

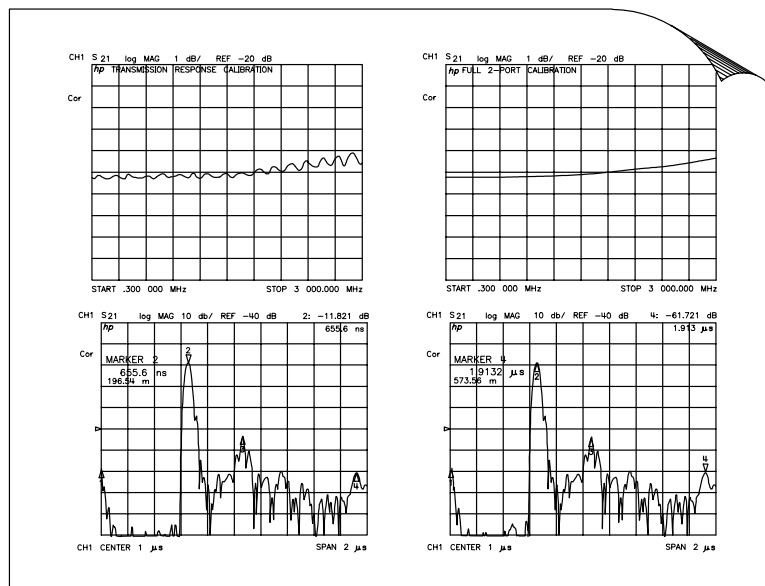


ph647c

To Plot Measurements in Page Quadrants

1. Define the plot, as explained in “Defining a Plot Function” on page 7-15.
2. Press **Copy**, **SEL QUAD []**.
3. Choose the quadrant where you want your displayed measurement to appear on the hardcopy. The selected quadrant appears in the brackets under **SEL QUAD []**.

Figure 7-10. Plot Quadrants



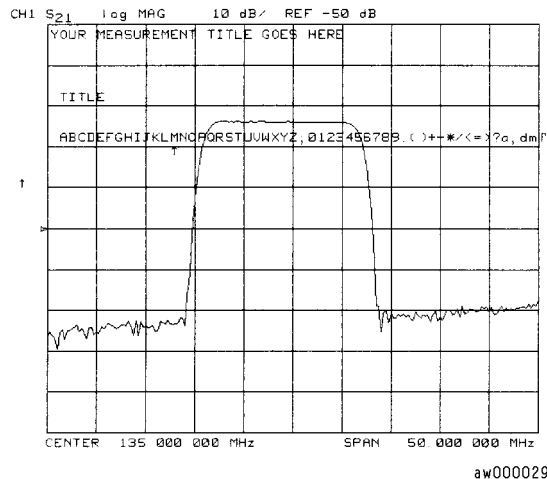
pg65e

4. Press **PLOT**. The analyzer assigns the first available default filename for the selected quadrant. For example, the analyzer would assign PLOT01LU if there were no other left-upper quadrant plots on the disk.
5. Make the next measurement that you want to see on your hardcopy.
6. Repeat this procedure for the remaining plot files that you want to see as quadrants on a page. If you want to see what quadrants you have already saved, press **Save/Recall**, to view the directory.

Titling the Displayed Measurement

1. Press **Display**, **MORE**, **TITLE**, to access the title menu.
2. Press **ERASE TITLE**, and enter the title you want for your measurement display.
 - If you have a DIN keyboard attached to the analyzer, type the title you want from the keyboard. Then press **ENTER**, to enter the title into the analyzer. You can enter a title that has a maximum of 50 characters. (For more information on using a keyboard with the analyzer, refer to the “Options and Accessories” chapter of the reference guide.)
 - If you do not have a DIN keyboard attached to the analyzer, enter the title from the analyzer front panel.
 - a. Turn the front panel knob to move the arrow pointer to the first character of the title.
 - b. Press **SELECT LETTER**.
 - c. Repeat the previous two steps to enter the rest of the characters in your title. You can enter a title that has a maximum of 50 characters.
 - d. Press **DONE**, to complete the title entry.

Figure 7-11. Example of a Display Title



CAUTION The **NEWLINE**, and **FORMFEED**, keys are not intended for creating display titles. Those keys are for creating commands to send to peripherals during a sequence program.

Configuring the Analyzer to Produce a Time Stamp

You can set a clock, and then activate it, if you want the time and date to appear on your hardcopies.

1. Press **System**, **SET CLOCK**.
2. Press **SET YEAR**, and enter the current year, followed by **x1**.
3. Press **SET MONTH**, and enter the current month of the year, followed **x1**.
4. Press **SET DAY**, and enter the current day of the month, followed by **x1**.
5. Press **SET HOUR**, and enter the current hour of the day (0-23), followed by **x1**.
6. Press **SET MINUTES**, and enter the next immediate minute, followed by **x1**.
7. Press **ROUND SECONDS**, when the current time is exactly as you have set it.
8. Press **TIME STAMP**, until **TIME STAMP ON**, appears on the softkey label.

Aborting a Print or Plot Process

1. Press the **Local**, key to stop all data transfer.
2. If your peripheral is not responding, press **Local**, again or reset the peripheral.

Printing or Plotting the List Values or Operating Parameters

Press **Copy**, **LIST**, and select the information that you want to appear on your hardcopy.

- Choose **LIST VALUES**, if you want a tabular listing of the measured data points, and their current values, to appear on your hardcopy. This list will also include the limit test information, if you have the limits function activated.
- Choose **OP PARMS (MKRS etc)**, if you want a tabular listing of the parameters for both measurement channels to appear on your hardcopy. The parameters include: operating parameters, marker parameters, and system parameters that relate to the control of peripheral devices.

If You Want a Single Page of Values

1. Choose **PRINT MONOCHROME**, for a printer or **PLOT**, for a plotter peripheral, to create a hardcopy of the displayed page of listed values.
2. Press **NEXT PAGE**, to display the next page of listed values. Press **PREVIOUS PAGE**, to display the previous page of listed values. Or, you can press **NEXT PAGE**, or **PREVIOUS PAGE**, repeatedly to display a particular page of listed values that you want to appear on your hardcopy. Then repeat the previous step to create the hardcopy.
3. Repeat the previous two steps until you have created hardcopies for all the desired pages of listed values.

If you are printing the list of measurement data points, each page contains 30 lines of data. The number of pages is determined by the number of measurement points that you have selected.

If You Want the Entire List of Values

Choose **PRINT ALL**, to print all pages of the listed values.

NOTE If you are printing the list of operating parameters, only the first four pages are printed. The fifth page, system parameters, is printed by displaying that page and then pressing **PRINT**.

Solving Problems with Printing or Plotting

If you encounter a problem when you are printing or plotting, check the following list for possible causes:

- Look in the analyzer display message area. The analyzer may show a message that will identify the problem. Refer to the "Error Messages" chapter of the reference guide if a message appears.
- If necessary, refer to the peripheral configuration procedures in this chapter to check that you have done the following:
 - connected an interface cable between the peripheral and the analyzer
 - connected the peripheral to ac power
 - switched on the power
 - switched the peripheral on line
 - selected the correct printer or plotter type
 - selected the correct interface (GPIB, parallel, serial)
- If you are using a laser printer for plotting, and the printer is outputting partial plots, the printer may require more memory or the page protection activated.

NOTE Consult your printer documentation for information on upgrading memory and how to activate page protection.

- **For GPIB peripherals:** Make sure that the analyzer address setting for the peripheral corresponds to the actual GPIB address of the peripheral. The procedure is explained earlier in this chapter.
- **For GPIB peripherals:** Make sure that the analyzer is in system controller mode, by pressing **Local**, **SYSTEM CONTROLLER**, if the analyzer is not connected to an external controller. Otherwise, the analyzer must be in the pass control mode.
- Substitute the interface cable.
- Substitute a different printer or plotter.

Numerics

4 Param Displays softkey, [2-12](#)

A

aborting a print or plot process, [7-34](#)

absolute

ripple test value, [2-57](#), [2-58](#)

activating

averaging, [3-8](#)

display markers, [2-18](#)

fixed markers, [2-22](#)

limit test, [2-48](#)

activating the bandwidth test, [2-62](#)

active channel display, [2-5](#)

adjusting

display colors, [2-15](#)

Agilent offices, [5-8](#)

amplitude, tracking, [2-35](#)

analyzer internal memory, what you can save, [6-3](#)

ASCII data formats, [6-8](#)

CITIfile, [6-8](#)

S2P data format, [6-9](#)

auto-feed, selecting, [7-15](#)

autostarting sequences, [2-74](#)

averaging, activating, [3-8](#)

B

band

adding a frequency, [2-53](#)

changing a frequency, [2-53](#)

deleting a frequency, [2-54](#)

frequency, [2-51](#)

bandwidth, [2-61](#)

bandwidth markers

displaying, [2-63](#)

bandwidth test

activating, [2-62](#)

running, [2-62](#)–[2-65](#)

setting up limits, [2-60](#)–[2-62](#)

bandwidth testing, [2-60](#)–[2-65](#)

bandwidth value

displaying, [2-64](#)

bandwidth, searching for, [2-34](#)

basic measurements, making, [2-3](#)

blanking the display, [2-14](#)

C

cables, interconnecting, [3-4](#)

calculating statistics of measurement data, [2-35](#)

calibration

electrical, [4-10](#)

O/O response and isolation, [4-11](#)

optical (O/E), [4-10](#)

optical (O/O), [4-10](#)

procedures, [4-10](#)

calibration considerations, [4-3](#)

calibration standards, [4-4](#)

clarifying type-N connector sex, [4-3](#)

device measurements, [4-3](#)

error-correction stimulus state, [4-8](#)

frequency response of calibration standards, [4-4](#)

interpolated error correction, [4-7](#)

measurement parameters, [4-3](#)

omitting isolation calibration, [4-3](#)

restarting a calibration, [4-4](#)

saving calibration data, [4-4](#)

calibration data, saving, [4-4](#), [4-38](#)

calibration kit menu, [4-30](#)

defining standard menus, [4-31](#)

label class menu, [4-37](#)

label kit menu, [4-38](#)

label standard menu, [4-35](#)

specify class menu, [4-35](#)

specifying offset menu, [4-33](#)

calibration kit, saving a modified, [4-38](#)

calibration standards, [4-4](#)

calibration techniques

improper, [3-4](#)

capacitance, fringe, [4-5](#)

center frequency, setting, [2-28](#)

changing

sequence title, [2-70](#)

system bandwidth, [3-8](#)

changing the ripple limits color, [2-57](#)

channel position softkey, [2-11](#)

characteristics of the filter, [2-38](#)

choosing

display elements, [7-15](#)

plot speed, [7-18](#)

scale, [7-17](#)

CITIfile, [6-8](#)

clarifying type-N connector sex, [4-3](#)

cleaning

adapters, [1-10](#)

fiber-optic connections, [1-2](#), [1-9](#)

non-lensed connectors, [1-9](#)

clearing a sequence from memory, [2-70](#)

color

changing the ripple limits, [2-57](#)

ripple test message, [2-55](#)

color printer, using, [7-6](#)

colors of the display, adjusting, [2-15](#)

default, [2-15](#)

intensity, [2-15](#)

modified, [2-16](#)

modify colors menu, [2-15](#)

command

deleting, [2-68](#)

inserting, [2-69](#)

modifying, [2-69](#)

commands that require clean sweep, [2-73](#)

commands that sequencing completes before next command, [2-73](#)

comma-separated values. See CSV format

compressed dust remover, [1-8](#)

computer, what you can save, [6-4](#)

configuring

plot function, [7-10](#)

connections

care manual, [1-11](#)

care of, [1-2](#)

connector

care, [3-3](#)

repeatability, [3-4](#)

continuous markers, [2-17](#)

cotton swabs, [1-8](#)

coupling display markers, [2-24](#)

creating

flat limit lines, [2-42](#)

sequence, [2-66](#)

sloping limit line, [2-45](#)

creating single point limits, [2-46](#)

CSV format, saving measurement results in, [6-11](#)

CW frequency, setting, [2-31](#)

D

data formats, ASCII, [6-8](#)

data trace, [2-14](#)

saving to display memory, [2-14](#)

decision making functions, [2-75](#)

decoupled

channel power, [2-7](#)

stimulus, [2-7](#)

default colors, [2-15](#)

defining

plot function, [7-15](#)

print function, [7-6](#)

defining a print function

color printer, using, [7-6](#)

resetting print parameters to default value, [7-7](#)

deleting

commands, [2-68](#)

line segments, [2-48](#)

deleting a file, [6-19](#)

all files, [6-19](#)

instrument state file, [6-19](#)

delta markers, [2-21](#)

device measurements, [4-3](#)

discrete markers, [2-17](#)

disk

formatting, [6-22](#)

disk, plotting a measurement to, [7-13](#)

- display elements, choosing, 7-15
- display functions, 2-4
 - active channel display, 2-5
 - titling, 2-5
 - adjusting colors of the display, 2-15
 - blanking the display, 2-14
 - data trace
 - saving to display memory, 2-14
 - four-channel display
 - 4 Param Displays softkey, 2-12
 - Channel Position softkey, 2-11
 - customizing, 2-11
 - viewing, 2-8
 - measurement data
 - dividing by the memory trace, 2-14
 - viewing, 2-14
 - memory math functions, 2-13
 - memory trace
 - viewing, 2-14
 - memory traces, 2-13
 - primary measurement
 - dual channel mode with decoupled channel power, 2-7
 - dual channel mode with decoupled stimulus, 2-7
 - primary measurement channels, 2-6
 - viewing, 2-6
- display intensity, 2-15
- display markers
 - coupling, 2-24
 - uncoupling, 2-24
- display markers, activating, 2-18
- display memory, 2-14
- display reference value, setting, 2-29
- display ripple test limits, 2-56
- display ripple test values, 2-57
- displayed measurement, titling, 7-32
- displaying the bandwidth markers, 2-63
- displaying the bandwidth value, 2-64
- drift, frequency, 3-5
- drift, temperature, 3-4
- dual channel mode with decoupled power, 2-7
- dual channel mode with decoupled stimulus, 2-7
- dust caps, 1-8
- dynamic range, increasing, 3-7
- E**
 - editing a sequence, 2-68
 - deleting commands, 2-68
 - editing line segments, 2-47
 - editing ripple limits, 2-53–2-54
 - electrical
 - calibrations, 4-10
 - offset, 4-5
- electrical delay, setting, 2-30
- electrostatic discharge, 1-12
- embedding loop counter value in title, 2-74
- error correcting measurements, 4-9
- error-correction
 - stimulus state, 4-8
- ESD
 - reducing damage caused by ESD, 1-13
 - static-safe work station, 1-12
- exit HPGL mode, 7-25
- sending to the printer, 7-25
- external
 - disk drive, 6-23
- F**
 - features that operate differently in sequence, 2-72
 - fiber optics
 - cleaning connections, 1-2
 - file
 - deleting, 6-19
 - index numbers, 6-12
 - recalling, 6-21
 - renaming, 6-20
 - sequential CSV naming of, 6-12
 - to delete all, 6-19
 - filter, characteristics, 2-38
 - fixed markers, 2-22
 - flat limit lines, 2-42
 - floppy disk, what you can save, 6-4
 - foam swabs, 1-8
 - form feed sequence, 7-25
 - sending to the printer, 7-25
 - format markers, polar, 2-25
 - formatting a disk, 6-22
 - forward stepping in edit mode, 2-73
 - four-channel display
 - 4 Param Displays softkey, 2-12
 - Channel Position softkey, 2-11
 - customizing, 2-11
 - viewing, 2-8
 - frequency
 - span, setting, 2-29
 - frequency bands
 - adding, 2-53
 - changing, 2-53
 - deleting, 2-54
 - setting, 2-51
 - frequency drift, 3-5
 - frequency response of calibration
 - standards, 4-4
 - electrical offset, 4-5
 - fringe capacitance, 4-5
 - fringe capacitance, 4-5
- G**
 - generating files in a loop counter
 - example, 2-79
 - gosub sequence command, 2-74
 - graphic files
 - saving measurement results as, 6-13
 - grids
 - moving marker information off the grids, 2-19
- H**
 - HPGL
 - compatible printer, 7-21
 - initialization sequence, sending to the printer, 7-25
 - initialization sequence, storing, 7-24
 - HPGL compatible printer, 7-24
 - HPGL/2 compatible printer, 7-10
- I**
 - increasing dynamic range, 3-7
 - increasing test port input power, 3-7
 - reducing receiver crosstalk, 3-7
 - reducing the receiver noise floor, 3-7
 - increasing measurement accuracy, 3-4
 - connector repeatability, 3-4
 - frequency drift, 3-5
 - interconnecting cables, 3-4
 - performance verification, 3-5
 - reference plane and port extensions, 3-5
 - temperature drift, 3-4
 - increasing test port input power, 3-7
 - in-depth sequencing information, 2-72
 - autostarting sequences, 2-74
 - commands completed before next sequence begins, 2-72
 - commands that require a clean sweep, 2-73
 - decision making functions, 2-75
 - embedding loop counter value in title, 2-74
 - features that operate differently in a sequence, 2-72
 - forward stepping in edit mode, 2-73
 - gosub sequence command, 2-74
 - limit test decision making, 2-75
 - loop counter decision making, 2-75
 - sequence decision making menu, 2-75
 - sequence size, 2-73
 - sequence that jumps to itself, 2-75
 - sequencing special functions menu, 2-75
 - titles, 2-73
 - TTL input decision making, 2-75

Index

- TTL out menu, [2-75](#)
- input connector, [1-2](#)
- inserting a command, [2-69](#)
- instrument state
 - file, deleting, [6-19](#)
 - files, [6-14](#)
 - re-saving, [6-18](#)
 - saving, [4-38](#), [6-5](#)
 - saving and recalling, [6-3](#)
- interconnecting cables, [3-4](#)
- internal memory, [6-3](#)
- interpolated error correction, [4-7](#)
- isolation
 - calibration, omitting, [4-3](#)
- J**
- jpeg files, saving results as, [6-13](#)
- L**
- limit lines
 - creating flat limit lines, [2-42](#)
 - creating single point limits, [2-46](#)
 - editing line segments, [2-47](#)
 - measurement parameters, [2-42](#)
 - offsetting limit lines, [2-49](#)
 - running a limit test, [2-48](#)
 - sloping limit line, [2-45](#)
 - using to test a device, [2-42](#)
- limit test
 - decision making, [2-75](#)
 - example sequence, [2-81](#)
- limit test, running, [2-48](#)
 - activating the limit test, [2-48](#)
 - reviewing the limit line segments, [2-48](#)
- line segments, editing, [2-47](#)
 - deleting line segments, [2-48](#)
- line types, selecting, [7-17](#)
- list values, printing or plotting, [7-35](#)
- loading a sequence from a disk, [2-72](#)
- loop counter
 - decision making, [2-75](#)
 - example sequence, [2-78](#)
- lower stopband parameters, [2-39](#)
- M**
- making a basic measurement, [2-3](#)
- margin
 - ripple test value, [2-57](#), [2-58](#)
- markers, [2-17](#)
 - calculating statistics of measurement data, [2-35](#)
 - continuous, [2-17](#)
 - coupling display markers, [2-24](#)
 - CW frequency, setting, [2-31](#)
 - delta markers, [2-21](#)
 - discrete, [2-17](#)
 - display markers
 - activating, [2-18](#)
 - fixed marker
 - activating, [2-22](#)
 - moving marker information off of the grids, [2-19](#)
 - polar format markers, [2-25](#)
 - setting measurement parameters, [2-27](#)
 - smith chart markers, [2-26](#)
 - specific amplitude, searching for, [2-32](#)
 - uncoupling display markers, [2-24](#)
 - maximum amplitude, searching for, [2-32](#)
- maximum and minimum, [2-61](#)
- maximum bandwidth, [2-61](#)
- measurement
 - accuracy, increasing, [3-4](#)
 - plotting to a disk, [7-13](#)
 - printing or plotting results, [7-3](#)
 - results, saving, [6-6](#)
 - results, saving graphically, [6-13](#)
- measurement data, [2-14](#)
 - dividing by the memory trace, [2-14](#)
 - viewing, [2-14](#)
- measurement parameters, [2-38](#), [4-3](#)
 - center frequency, setting, [2-28](#)
 - display reference value, setting, [2-29](#)
 - electrical delay, setting, [2-30](#)
 - frequency span, setting, [2-29](#)
 - lower stopband parameters, [2-39](#)
 - markers, setting with, [2-27](#)
 - passband parameters, [2-39](#)
 - start frequency, setting, [2-27](#)
 - stop frequency, setting, [2-28](#)
 - upper stopband parameters, [2-39](#)
- measurements
 - basic, [2-3](#)
- memory math functions, [2-13](#)
- memory trace, [2-13](#), [2-14](#)
 - viewing, [2-14](#)
- menu
 - calibration kit, [4-30](#)
- microwave connector care, [3-3](#)
- minimum amplitude, searching for, [2-32](#)
- minimum bandwidth, [2-61](#)
- modified colors, [2-16](#)
 - recalling, [2-16](#)
 - saving, [2-16](#)
- modify colors menu, [2-15](#)
- modifying
 - command, [2-69](#)
- modifying calibration kits
 - calibration kit menu, [4-30](#)
- saving to a disk, [4-38](#)
- verifying performance, [4-39](#)
- multiple
 - measurements per page, plotting from a disk, [7-28](#)
 - measurements, plotting on a full page, [7-29](#)
 - plots, outputting to a single page using a plotter, [7-27](#)
- N**
- N dB Point, [2-61](#)
- names for CSV files, [6-12](#)
- naming files generated by a sequence, [2-71](#)
- O**
- O/O
 - response and isolation bandwidth calibration, [4-11](#)
- offset, electrical, [4-5](#)
- offsetting limit lines, [2-49](#)
- omitting isolation calibration, [4-3](#)
- operating parameters, printing or plotting, [7-35](#)
- optical
 - calibrations, [4-10](#)
- outputting
 - multiple plots to a single page using a plotter, [7-27](#)
 - plot files, [7-14](#)
 - plot files from a PC to a plotter, [7-23](#)
 - single page plots using a printer, [7-26](#)
 - outputting plot files from a PC to an HPGL compatible printer, [7-24](#)
 - sending the exit HPGL mode and form feed sequence to the printer, [7-25](#)
 - sending the HPGL initialization sequence to the printer, [7-25](#)
 - sending the plot file to the printer, [7-25](#)
 - storing the exit HPGL mode and form feed sequence, [7-25](#)
 - storing the HPGL initialization sequence, [7-24](#)
- P**
- page quadrants, plotting measurements in, [7-30](#)
- parameters
 - lower stopband, [2-39](#)
 - measurement, [2-38](#), [4-3](#)
 - passband, [2-39](#)
 - upper stopband, [2-39](#)
- passband parameters, [2-39](#)

- PC, to view files on, 7-22
- pen numbers and colors, selecting, 7-16
- pen plotter, 7-19, 7-20
 - plotting to, 7-11
- performance verification, 3-5
- performance, verifying, 4-39
- places where you can save, 6-3
- plot
 - aborting a process, 7-34
- plot files
 - outputting from a PC to a plotter, 7-23
 - outputting from a PC to an HPGL compatible printer, 7-24
 - sending to the printer, 7-25
 - to output, 7-14
 - to view on a PC, 7-22
- plot function, configuring, 7-10
 - plotting to a pen plotter, 7-11
 - plotting to an HPGL/2 compatible printer, 7-10
- plot function, defining, 7-15
 - choosing display elements, 7-15
 - choosing plot speed, 7-18
 - choosing scale, 7-17
 - resetting plotting parameters to
 - default values, 7-18
 - selecting auto-feed, 7-15
 - selecting line types, 7-17
 - selecting pen numbers and colors, 7-16
- plot speed, choosing, 7-18
- plotting
 - measurement results, 7-3
 - measurement to a disk, 7-13
 - measurements in page quadrants, 7-30
 - multiple measurements on a full page, 7-29
 - multiple measurements per page from a disk, 7-28
 - multiple measurements per page using a pen plotter, 7-20
 - one measurements per page using a pen plotter, 7-19
 - parameters, resetting to default values, 7-18
 - plotting to an HPGL compatible printer, 7-21
 - solving problems, 7-36
- plotting a measurement to a disk to output the plot files, 7-14
- polar format markers, 2-25
- port extensions, 3-5
- primary measurement channels, viewing, 2-6
- print
 - aborting a process, 7-34
- print function
 - configuring, 7-4
 - defining, 7-6
- printer
 - color printer, using, 7-6
 - HPGL compatible printer, 7-21, 7-24
 - HPGL/2 compatible printer, 7-10
- printing
 - measurement results, 7-3
 - multiple measurements per page, 7-9
 - one measurement per page, 7-8
 - parameters, resetting to default values, 7-7
 - sequence, 2-72
 - solving problems, 7-36
- printing or plotting the list values or
 - operating parameters, 7-35
 - entire list of values, 7-35
 - single page of values, 7-35
- procedures for error correcting measurements, 4-9
 - types of error correction, 4-9
- purging a sequence from a disk, 2-72
- R**
 - recalling a file, 6-21
 - solving problems, 6-23
 - recalling instrument states, 6-3
 - receiver
 - crosstalk, reducing, 3-7, 3-9
 - noise floor, reducing, 3-7
 - reducing trace noise, 3-8
 - activating averaging, 3-8
 - changing system bandwidth, 3-8
 - reference plane and port extensions, 3-5
 - renaming a file, 6-20
 - repeatability, connector, 3-4
 - re-saving an instrument state, 6-18
 - resetting
 - plotting parameters to default values, 7-18
 - printing parameters to default values, 7-7
 - restarting a calibration, 4-4
 - return loss, 1-7
 - reviewing the limit line segments, 2-48
 - ripple limit testing, 2-50–2-59
 - ripple limits
 - editing, 2-53–2-54
 - running the test, 2-55–2-59
 - setting, 2-50–2-52
 - ripple test
 - absolute value, 2-58
 - displaying limits, 2-56
 - displaying values, 2-57
 - frequency bands, 2-51
 - margin value, 2-58
 - message color, 2-55
 - starting and stopping, 2-55
 - running a bandwidth test, 2-62–2-65
 - running a limit test, 2-48
 - running a sequence, 2-68
 - running the ripple limits test, 2-55–2-59
- S**
 - S2P data format, 6-9
 - sales and service offices, 5-8
 - saving
 - calibration data, 4-4, 4-38
 - data trace, 2-14
 - instrument state, 4-38, 6-5
 - measurement results, 6-6
 - measurement results graphically, 6-13
 - modified calibration kits, 4-38
 - to a disk, 4-38
 - saving a file
 - solving problems, 6-23
 - saving and recalling instrument states, 6-3
 - places where you can save, 6-3
 - what you can save to a computer, 6-4
 - what you can save to a floppy disk, 6-4
 - what you can save to the analyzer's internal memory, 6-3
 - saving measurement results
 - ASCII data formats, 6-8
 - instrument state files, 6-14
 - scale, choosing, 7-17
 - searching for specific amplitude, 2-32
 - selecting
 - auto-feed, 7-15
 - line types, 7-17
 - pen numbers and colors, 7-16
 - sending the exit HPGL mode and form feed sequence to the printer, 7-25
 - sending the HPGL initialization sequence to the printer, 7-25
 - sending the plot file to the printer, 7-25
 - sequence
 - changing the title, 2-70
 - clearing from memory, 2-70
 - creating, 2-66
 - decision making menu, 2-75
 - editing, 2-68
 - generating files in a loop counter example, 2-79
 - in-depth information, 2-72

- jumps to itself, [2-75](#)
- limit test example, [2-81](#)
- loading from a disk, [2-72](#)
- loop counter example, [2-78](#)
- naming files, [2-71](#)
- printing, [2-72](#)
- purging from a disk, [2-72](#)
- running, [2-68](#)
- size, [2-73](#)
- stopping, [2-68](#)
- storing to disk, [2-71](#)
- sequencing
 - special functions menu, [2-75](#)
 - test, [2-66](#)
- service
 - sales and service offices, [5-8](#)
- setting ripple limits, [2-50–2-52](#)
- setting up bandwidth test limits, [2-60–2-62](#)
- single page plots, outputting using a printer, [7-26](#)
- single point limits, [2-46](#)
- sloping limit line, [2-45](#)
- smith chart markers, [2-26](#)
- softkey
 - 4 Param Displays, [2-12](#)
 - channel position, [2-11](#)
- solving problems with printing or plotting, [7-36](#)
- solving problems with saving or recalling files, [6-23](#)
 - using an external disk drive, [6-23](#)
- specific amplitude, [2-32](#)
 - bandwidth, searching for, [2-34](#)
 - maximum amplitude, searching for, [2-32](#)
 - minimum amplitude, searching for, [2-32](#)
 - target amplitude, searching for, [2-33](#)
 - tracking the amplitude, [2-35](#)
- spreadsheet, saving test file for a, [6-11](#)
- standards, calibration, [4-4](#)
- start frequency, setting, [2-27](#)
- starting the ripple test, [2-55](#)
- statistics of measurement data, calculating, [2-35](#)
- stepped list mode, [2-37](#)
- stimulus state, error-correction, [4-8](#)
- stop frequency, setting, [2-28](#)
- stopping a sequence, [2-68](#)
- stopping the ripple test, [2-55](#)
- storing
 - exit HPGL mode and form feed sequence, [7-25](#)
 - HPGL initialization sequence, [7-24](#)
 - sequence on a disk, [2-71](#)
- swabs, [1-8](#)
- swept list mode
 - calibrate, [2-40](#)
 - characteristics of the filter, [2-38](#)
 - device under test, connect, [2-37](#)
 - measure, [2-40](#)
 - measurement parameters, [2-38](#)
 - stepped list mode, [2-37](#)
 - to test a device, [2-37](#)
- system
 - bandwidth, changing, [3-8](#)
- T**
- taking care of microwave connectors, [3-3](#)
- target amplitude, searching for, [2-33](#)
- temperature drift, [3-4](#)
- test
 - bandwidth, [2-60–2-65](#)
 - ripple limit, [2-50–2-59](#)
- test port input power, increasing, [3-7](#)
- test sequencing, [2-66](#)
 - changing the sequence title, [2-70](#)
 - clearing a sequence from memory, [2-70](#)
 - creating a sequence, [2-66](#)
 - editing a sequence, [2-68](#)
 - generating files in a loop counter example, [2-79](#)
 - in-depth sequencing information, [2-72](#)
 - inserting a command, [2-69](#)
 - limit test example sequence, [2-81](#)
 - loading a sequence from a disk, [2-72](#)
 - loop counter example sequence, [2-78](#)
 - modifying a command, [2-69](#)
 - naming files generated by a sequence, [2-71](#)
 - printing a sequence, [2-72](#)
 - purging a sequence from a disk, [2-72](#)
 - running a sequence, [2-68](#)
 - stopping a sequence, [2-68](#)
 - storing a sequence on a disk, [2-71](#)
 - using to test a device, [2-77](#)
- text file, saving measurements as a, [6-11](#)
- time stamp, [7-33](#)
- title, [2-73](#)
- title, display, [2-5](#)
- titling the displayed measurement, [7-32](#)
- to produce a time stamp, [7-33](#)
- trace noise, reducing, [3-8](#)
- tracking the amplitude, [2-35](#)
- TTL**
 - out menu, [2-75](#)
- type-N connector sex
 - clarifying, [4-3](#)
- types of error correction, [4-9](#)
- U**
- uncoupling display markers, [2-24](#)
- upper stopband parameters, [2-39](#)
- V**
- verification, performance, [3-5](#)
- verifying performance, [4-39](#)
- viewing plot files on a PC, [7-22](#)
- W**
- what you can save to a computer, [6-4](#)
- what you can save to a floppy disk, [6-4](#)
- what you can save to the analyzer's internal memory, [6-3](#)

