Agilent 4396B Network/Spectrum/Impedance Analyzer Task Reference

Manual Change

Agilent Part No. N/A

January 2007

Change 1

Change the equations for the noise level measurement (page 3-3) to the following.

Converting to a Different Equivalent Noise Bandwidth

1. Calculate the conversion factor using the following equations with displayed units:

Unit	Use
dBm/Hz	*K=10logBW*
dBV/√Hz、*dBμV/√Hz*	*K=10logBW*
W/Hz	*K=BW*
V/√Hz	*K=√BW*

Where, *BW* is the target equivalent noise bandwidth.

マニュアル チェンジ

変更 1

ノイズ・レベル測定（ページ 3-2）の計算式を以下に変更して下さい。

等価ノイズ・バンド幅の変換

1. 表示単位にあわせて、以下の式から変換係数 *K* を計算します。

表示単位	式
dBm/Hz	*K=10logBW*
dBV/√Hz、*dBμV/√Hz*	*K=10logBW*
W/Hz	*K=BW*
V/√Hz	*K=√BW*

ここで、*BW* は変換する等価ノイズ・バンド幅です。

Safety Summary

When you notice any of the unusual conditions listed below, immediately terminate operation and disconnect the power cable.
Contact your local Agilent Technologies sales representative or authorized service company for repair of the instrument. If you continue to operate without repairing the instrument, there is a potential fire or shock hazard for the operator.

- Instrument operates abnormally.
- Instrument emits abnormal noise, smell, smoke or a spark-like light during the operation.
- Instrument generates high temperature or electrical shock during operation.
- Power cable, plug, or receptacle on instrument is damaged.
- Foreign substance or liquid has fallen into the instrument.

Agilent 4396B Network/Spectrum/Impedance Analyzer

Task Reference

SERIAL NUMBERS

This manual applies directly to instruments with serial number prefix JP1KE.
For additional important information about serial numbers, read "Serial Number" in Appendix A.

Agilent Technologies

Agilent Part No. 04396-90040
Printed in Japan November 2002

Fourth Edition

Notice

The information contained in this document is subject to change without notice.

Agilent Technologies Japan, Ltd.
Component Test PGU-Kobe
1-3-2, Murotani, Nishi-ku, Kobe-shi,
Hyogo, 651-2241 Japan

Manual Printing History

The manual's printing date and part number indicate its current edition. The printing date changes when a new edition is printed. (Minor corrections and updates that are incorporated at reprint do not cause the date to change.) The manual part number changes when extensive technical changes are incorporated.

March 1997 ..... First Edition (part number: 04396-90020)

July 1997 ..... Second Edition (part number: 04396-90030)

March 2000 ..... Third Edition (part number: 04396-90030)

November 2002 ..... Fourth Edition (part number: 04396-90040)

Contents

4. Recommended Network Measurement Task Sequence

Figures

Tables

1

Overview

This chapter contains the following information:

- Key Operation Overview
- Numerical Entries
- Character Entries

After chapter 2, the procedural steps do not contain as much detailed information. Therefore, you should read this chapter to understand the basic rules of analyzer's operation.

Key Operation Overview

Hardkeys and Softkeys

There are two types of front-panel keys; hardkeys and softkeys. Most hardkeys that have a label display function menus on the right side of the display. These function menus are also called softkey menus.

Softkey menus list functions other than those accessed directly by the hardkeys. To activate a function that is on a softkey menu, press the key located to the right of the displayed function label.

This manual uses the following conventions:

(Hardkey)	Indicates a hardkey
`Softkey`	Indicates a softkey

Toggle Keys

Some softkeys toggle menu settings. On a softkey label, the currently active setting is displayed in high intensity and upper case characters. The inactive setting is displayed in low intensity and lower case characters.

This manual shows a toggled setting as follows:

- Toggle `DUAL CHAN on OFF` to `ON off`.

This line means that the dual channel function is currently OFF. You turn it ON by pressing the corresponding softkey. The resulting upper case expression, `ON`, shows that the function is now active (ON).

For more information about each hardkey and softkey, see the *Function Reference* manual.

Numerical Entries

A numerical data entry is preceded by some function settings. For example, before entering a center frequency you must specify the center frequency value. The analyzer provides the following three ways to enter numerical data:

Using	Description
Numerical keys [0] .. [9]	Enters numerical data directly and terminates the entry using the units terminator keys. This capability is useful when you know the value you must enter. For example, to enter 1 MHz, press [1] [M/μ].
[⇧] and [⇩] key	Increments or decrements the settings. This is useful for changing settings broadly.
Rotary knob 	Changes settings continuously. This is useful if you want to change the setting while verifying changes on the display.

Notes

You can use following techniques when entering numerical data:

- If you enter a wrong character, press [Back Space] to erase the last character entered.
- If you want to clear the current entry, press [Entry Off].

Character Entries

Save, recall, and display title functions require character entries. When a function that requires character entry is activated, a character entry menu is displayed (see Figure 1-1).

You can enter the characters by using following procedure:

To Enter Characters

1. Move the cursor ("↑") under the character you want to enter.
2. Press SELECT LETTER or [×1].
3. Repeat step 1 and 2 until all characters are entered.
4. Press DONE.

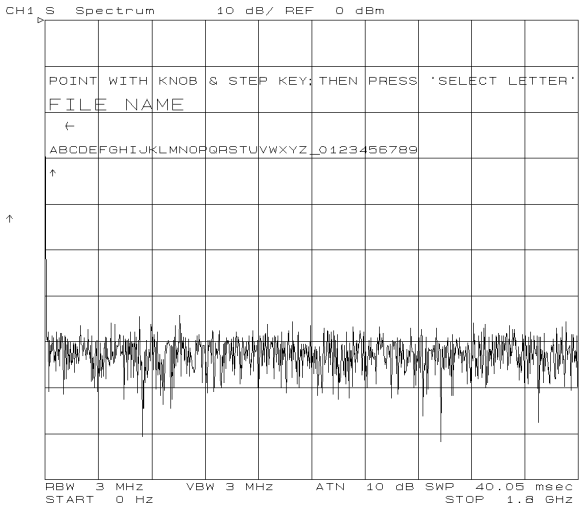


Figure 1-1. Character Entry Menu

You can use the following techniques when entering characters:

- If you want to use different case characters, press ⇧ and ⇩ to toggle the upper and lower case of an alphabet list:

  ABCDEFGHIJKLMNOPQRSTUVWXYZ_0123456789
  abcdefghijklmnopqrstuvwxyz_0123456789

- If you enter a wrong character, press BACK SPACE or Back Space to erase the last entered character.
- If you want to clear the current entry, press ERASE TITLE.

If the keyboard is connected, you can use it for the character entry. (Option 1C2 only)

2

Recommended Spectrum Measurement Task Sequence

This chapter describes a typical task sequence commonly used for any measurement using the spectrum analyzer mode.

If you are using the analyzer for the first time, see the *User's Guide* first. The *User's Guide* provides the information needed to install and set up the analyzer, and a quick start guide to introduce you to the analyzer.

The measurement tasks described in this chapter are as follows:

- Step 1: Preparing for a measurement
- Step 2: Setting the trigger
- Step 3: Selecting the measurement format
- Step 4: Setting the frequency range
- Step 5: Setting the vertical settings
- Step 6: Tuning the settings
- Step 7: Performing calibration
- Step 8: Reading the measured result

Generally, you can make a measurement by performing these steps. If you want to perform a more complex spectrum measurement, chapter 3 provides additional measurement techniques.

Step 1: Preparing for a Measurement

This step prepares the analyzer for a spectrum measurement. You must perform the following procedures before you enter measurement parameters (such as frequency range).

- To connect an unknown signal
- To preset the analyzer
- To select the active channel
- To select the spectrum analyzer mode

To Connect an Unknown Signal

- Connect the unknown signal to the S input on the front panel.

The S input is a standard N 50 Ω female connector. If you want to connect the BNC cable, use the N-BNC adapter (furnished).

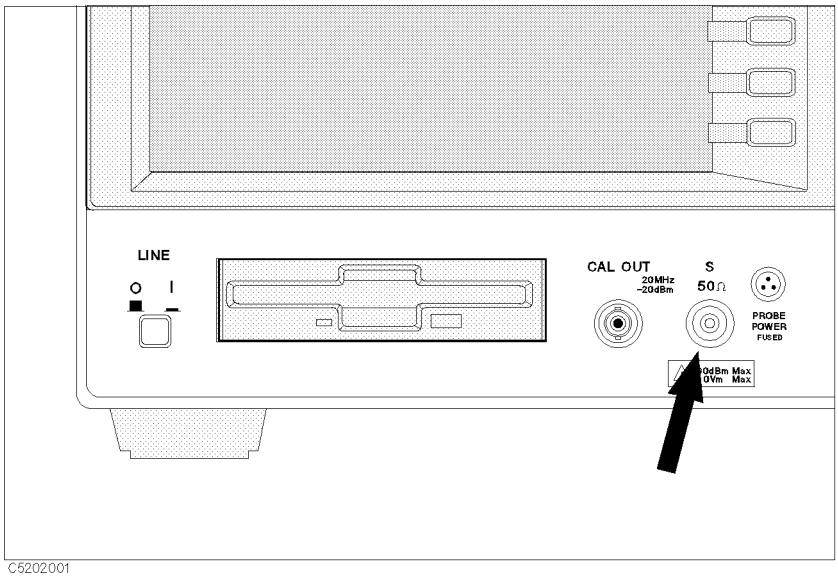


Figure 2-1. Location of the S Input

To Preset the Analyzer

- In the INSTRUMENT STATE block, press the green (Preset) key to set the analyzer to the preset state.

For additional information about the preset state, see appendix D of the *Function Reference*.

To Select the Active Channel

- In the ACTIVE CHANNEL block, press Chan 1 (channel 1) or Chan 2 (channel 2) to toggle the active channel.

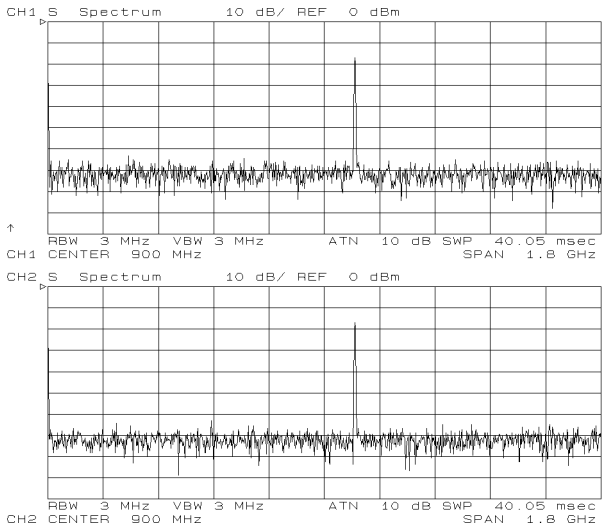
The analyzer has two independent channels. Each channel can have different settings that include whether the mode of operation is network analyzer or spectrum analyzer. Select the active channel before you select any other settings.

You can display two channels simultaneously.

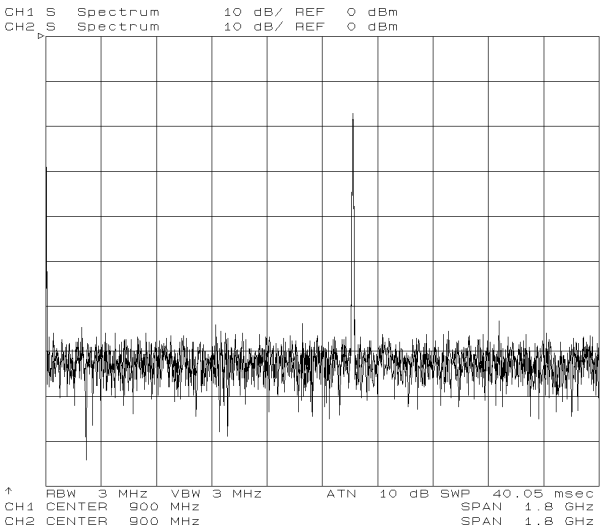
To Display Dual Channels

1. Press Display.
2. Toggle `DUAL CHAN on OFF` to `ON off`.
3. Press `MORE`.
4. Select the following display modes:

Display Mode	Toggle
Split channel	`SPLIT CHAN on OFF` to `ON off`
Overlap channel	`SPLIT CHAN ON off` to `on OFF`



Toggle to `SPLIT DISP ON off`.



Toggle to `SPLIT DISP on OFF`.

Figure 2-2. Dual Channel Display

To Select the Spectrum Analyzer Mode

1. Press Meas.
2. Press ANALYZER TYPE, SPECTRUM ANALYZER.

When you change the analyzer type (mode), the analyzer is reset to a known state. So, you must select the analyzer type before you select any additional settings.

Step 2: Setting the Trigger

This step selects the trigger source and the sweep condition using the following procedures:

- To select the trigger mode
  - To use the external trigger
- To select the sweep condition

To Select the Trigger Mode

1. Press (Trigger).
2. Press TRIGGER:[FREE RUN].
3. Select the trigger mode:

To Select a Trigger to	Press
Internal trigger source	FREE RUN
External trigger source	EXTERNAL [1]
Video trigger	VIDEO [2]
Manual	MANUAL
Time gated trigger	GATE [3]

1 See the "To Use the External Trigger" procedure.

2 The "To Stabilize the Trace Using the Video Trigger" in Chapter 3, describes how to use this trigger mode.

3 The "To Perform the Time Gated Spectrum Analysis" in Chapter 3, describes how to use this trigger mode. This function is option 1D6 only.

To Use the External Trigger

1. Connect the trigger source to the EXT TRIGGER connector on the rear panel of the analyzer.
2. Press (Trigger).
3. Press TRIGGER:[FREE RUN].
4. Press EXTERNAL.
5. Input a trigger signal to the analyzer.

The external trigger signal must be TTL level compatible.

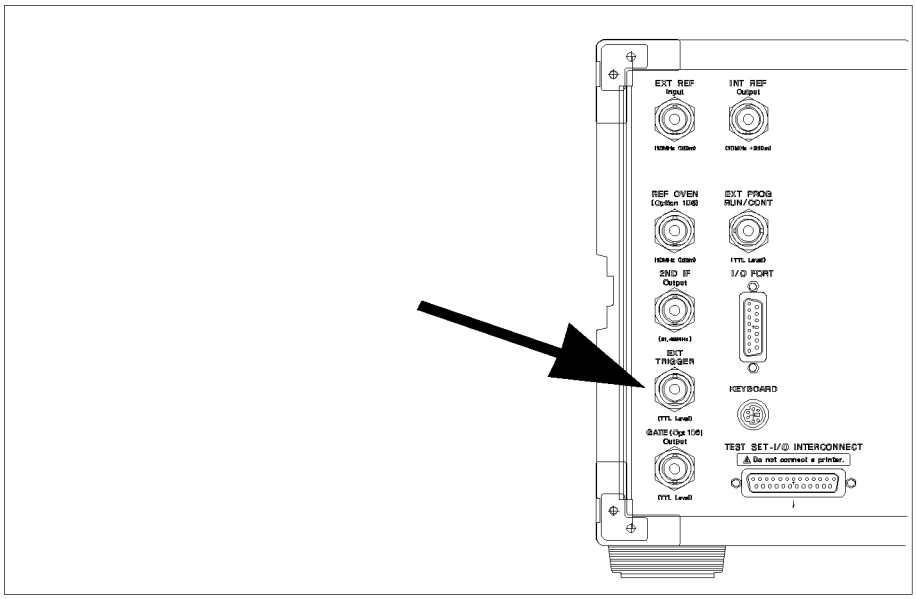


Figure 2-3. Location of EXT TRIGGER Connector

To Set the Trigger Signal Polarity.

1. Press (Trigger).
2. Press `TRIGGER:[FREE RUN]` .
3. Toggle `TRIG PLRTY POS neg` to `pos NEG` to set the trigger signal polarity to negative.

To Select the Sweep Condition

1. Press (Trigger).
2. Choose one of the following.

To Sweep	Do
Continuously	Press `CONTINUOUS` .
Single Time	Press `SINGLE` .
Specified Times	Press `NUMBER of GROUPS` . Then enter number of times to sweep.

Step 3: Selecting the Measurement Format

This step provides the following procedure:

- To select the display unit

To Select the Display Unit

1. Press (Format).
2. Select unit:

To Display	Unit	Press
Power	dBm	dBm
	W	WATT
Voltage	dBV	dBV
	dBμV	dBuV
	V	VOLT

You can change the displayed unit anytime you want. The analyzer calculates the unit conversion using the internal stored data. The sweep is not required (it can even be in the hold state).

Notes For more information about each unit, see chapter 5 of the *Function Reference*.

If you want to make a noise measurement instead of a spectrum measurement, see "To Measure the Noise Level" in Chapter 3.

Step 4: Setting the Frequency Range

The analyzer has some useful commands for setting the frequency range. This step provides the following procedures that are related to setting the frequency range.

- To set the center frequency
  - To set the marker position to center
  - To set the maximum peak to center
  - To change the center with the specified step size
- To set the frequency span
  - To set the frequency range to full span
  - To narrow the span setting
- To zoom to a part of the trace
  - To change the zooming magnification
  - To display a zoomed trace on the other channel
  - To zoom between the marker and the Δmarker

To Set the Center Frequency

1. Press (Center) to activate the center frequency function.
2. Change the center frequency to place the target signal in the center of the grid by using the following keys:

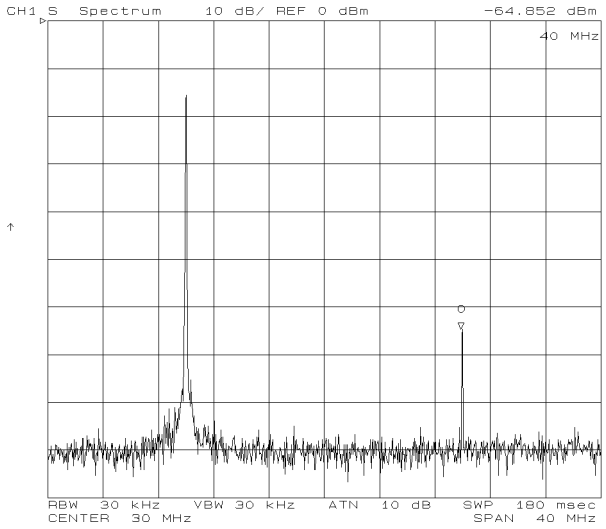
To	Use
Set directly	(0) ... (9) and units terminator keys
Change continuously	
Change with 1-2-5 steps[1]	(⇧) (⇩)

1 You can change the step size of (⇧) (⇩). See "To Change the Center with the Specified Step Size" in this step.

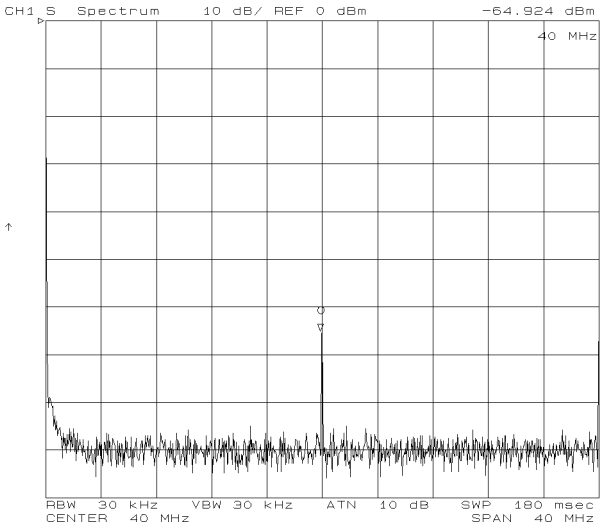
To Set the Marker Position to Center

1. Press (Marker→) to display the reverse-triangle shaped marker.
2. Place the marker on the position you want to set to the center by using the rotary knob.
3. Press `MKR→CENTER`.
4. Press (Entry Off).

The center frequency setting immediately changes to the marker position. If you are measuring an unknown signal, display the signal in full span first. Then move the signal to the center using this function.



Move the marker



Press MKR→CENTER

Figure 2-4. Marker to Center

To Set the Maximum Peak to Center

1. Press (Marker→).
2. Press PEAK→CENTER.
3. Press (Entry Off).

This function changes the center frequency to display the maximum peak in the center of the grid.

Note

When the frequency span setting is too wide, the peak cannot be placed in the center of the grid correctly. If the peak is not on the center of the grid, press PEAK→CENTER again.

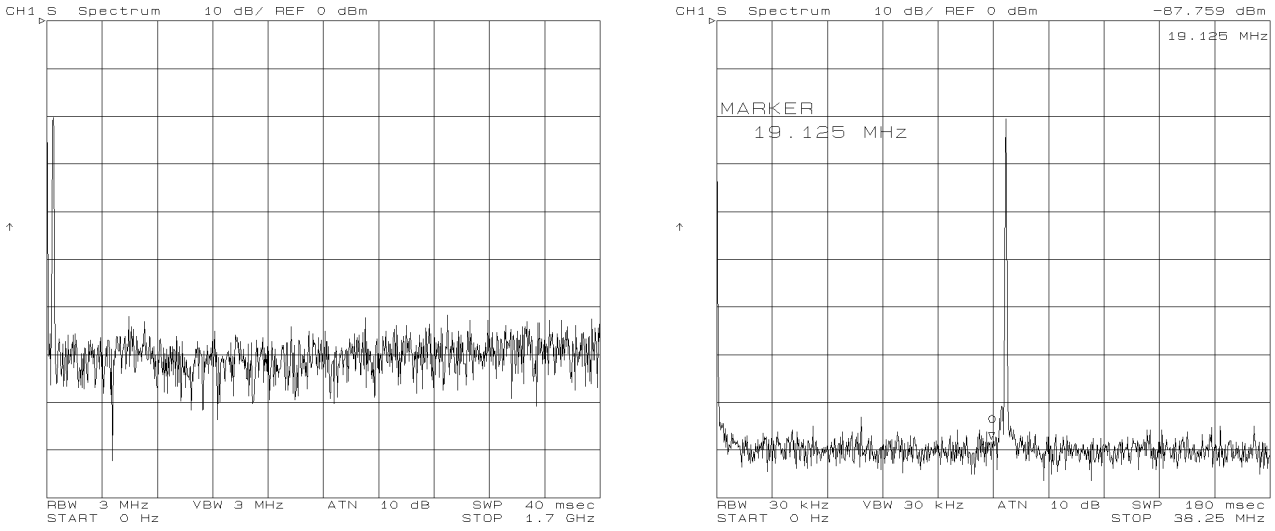


Figure 2-5. Peak to Center

To Change the Center with the Specified Step Size

1. Press (Center).
2. Do one of the following:
   - Press `CENTER STEP SIZE`. Then set a step size directly by using (0) ... (9) and the units terminator keys.
   - Press (Marker). Move the marker to the point you want to use as the step size frequency. Then press (Center) `MKR→CNTR STEP`.
3. Toggle `STEP SIZE AUTO man` to `auto MAN`.
4. Press (Center).
5. Press (⇧) to increment (or (⇩) to decrement) the center frequency setting the specified step size.

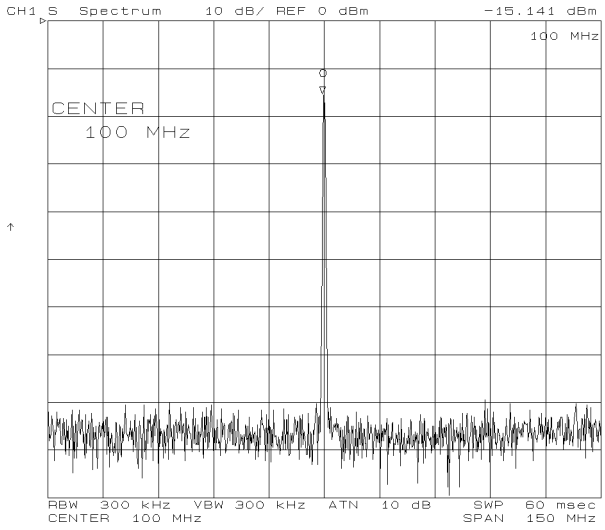
This function is useful to display peaks that have a constant interval (such as a harmonics) one after the other. The following example shows how to display the harmonics using this function.

Example: Displaying Harmonics. When you want to display the fundamental and the harmonics of 100 MHz signal:

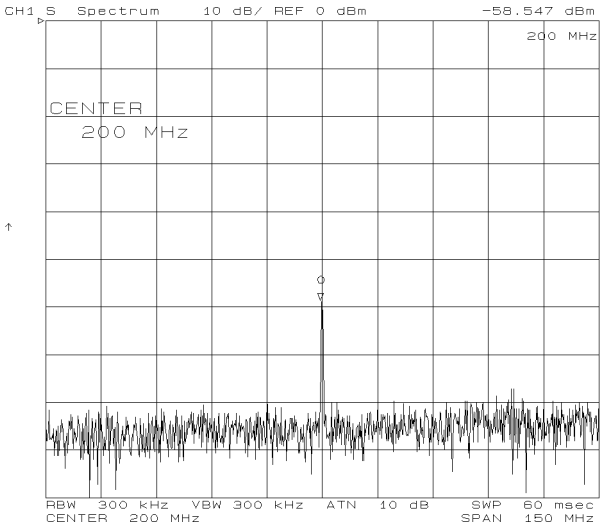
1. Press (Center) 100 (M/μ). Then set the span to display the fundamental in the center of the grid.
2. Press (Span) 150 (M/μ).
3. Toggle (Search) `SEARCH TRK on OFF` to `ON off` to enable the search track function.
4. Press `SEARCH:PEAK` to move the marker on the fundamental.
5. Press (Center) `MKR→CNTR STEP`. Enter 100 MHz (so the step size matches the fundamental frequency).

6. Toggle `STEP SIZE AUTO man` to `auto MAN` to enable the specified step size.
7. Press (Center). Then press (↑) to display the second harmonic.
8. To display higher order harmonics, press (↑) as required.

The marker searches for the next harmonic each time you change the center frequency by using the search track function.



Fundamental



Second Harmonics

Figure 2-6. Displaying Harmonics

To Set the Frequency Span

1. Press (Span).
2. Enter the frequency span to display the target peak in the optimum grid setting.

To	Use
Set directly	(0) ... (9) and units terminator keys
Change continuously	
Change with 1-2-5 steps	(↑) (↓)

To Set the Frequency Range to Full Span

1. Press (Span).
2. Press FULL SPAN

This function is useful when you want to get a general view of the spectrum after you have obtained the detailed view of a specific signal.

Notes You can set the sweep parameter using (Start) and (Stop) instead of (Center) and (Span). See "To Set the Sweep Parameter Using (Start) and (Stop)" in Chapter 4.

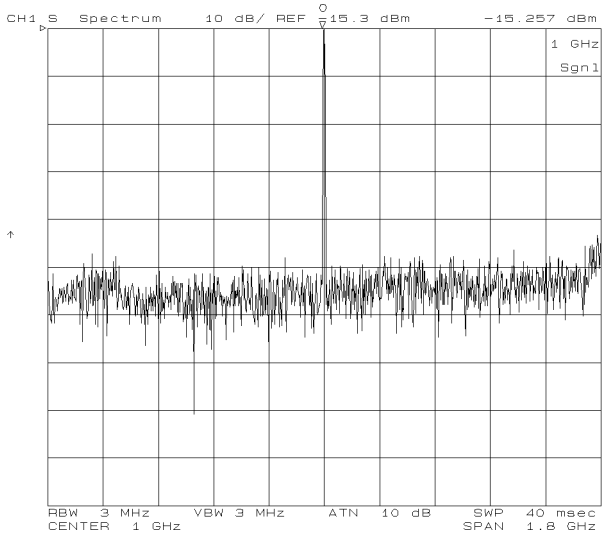
To Narrow the Span Setting

1. Press Search.
2. Press SEARCH:PEAK to place the marker on the carrier.
3. Toggle SIGNAL TRK on OFF to ON off.
4. Narrow the span setting. See the "To Set the Frequency Span" procedure.

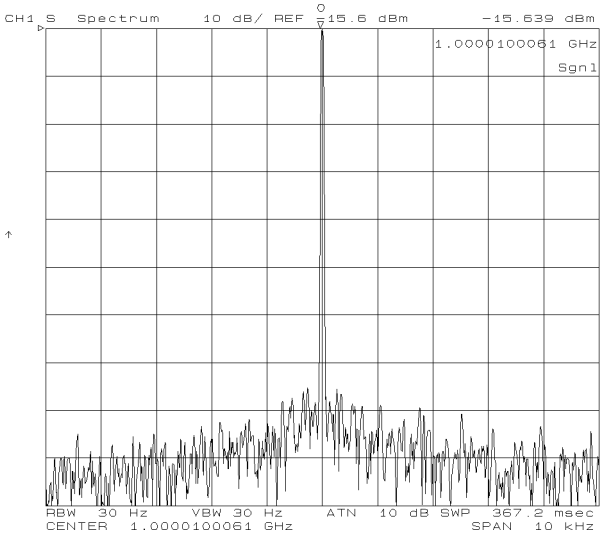
When you narrow the span setting substantially, the test signal can disappear from display. This happens because of the difference between the displayed and actual frequencies. For example, when the span setting is set to full span, the displayed test signal frequency has an error of approximately 2 MHz because of its resolution (1.8 GHz/800). If you narrow the span setting to less than the error frequency, the test signal can be lost from the display.

The signal track function allows you to avoid this situation. When signal track is enabled, the analyzer narrows the span setting while centering the test signal as you narrow the span setting. Therefore, the test signal is placed in the center of the grid.

The following figures show an example of narrowing the span with the signal track function. The actual signal frequency is 1.00001 GHz. When the center is fixed and the span is 10 kHz, the signal is out of display. The signal track function tracks the signal by changing the center frequency, and keeps displaying the signal in the center of the display.



Span 1.8 GHz



Span 10 kHz

Figure 2-7. Narrowing Span with Signal Track

To Zoom To a Part of the Trace

1. Move the marker to the point where you want to observe the signal details.
2. Press (Marker→).
3. Press MKR ZOOM.
4. To zoom more, press MKR ZOOM again.

To Change the Zooming Magnification

1. Press MORE ZOOMING APERTURE.
2. Enter a zooming aperture value as a percentage of the span.

If you want to magnify the display 20 times;, enter 5% for the zooming magnification.

To Display a Zoomed Trace on the Other Channel

1. Display two channels on the CRT. See the "To Display Dual Channels" procedure.
2. Toggle (Marker→) CROSS CHAN on OFF to ON off.
3. Press MKR ZOOM.

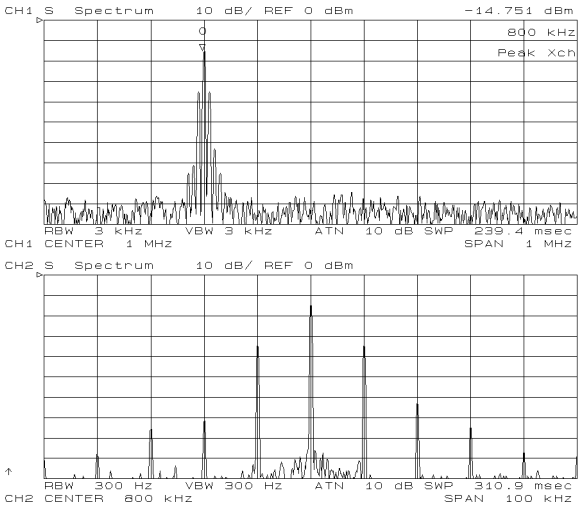


Figure 2-8. Marker Zoom

To Zoom Between the Marker and the ΔMarker

1. Press (Marker).
2. Move the marker to the start point you want to zoom to by using the  .
3. Press ΔMODE MENU ΔMKR.
4. Move the marker to the end point you want to zoom to by using the  .
5. Press (Marker→) MORE.
6. Press MKRΔ→SPAN.

The center frequency and span setting are changed automatically to display the specified area. You can display the zooming result on the other channel by toggling CROSS CHAN on OFF to ON off.

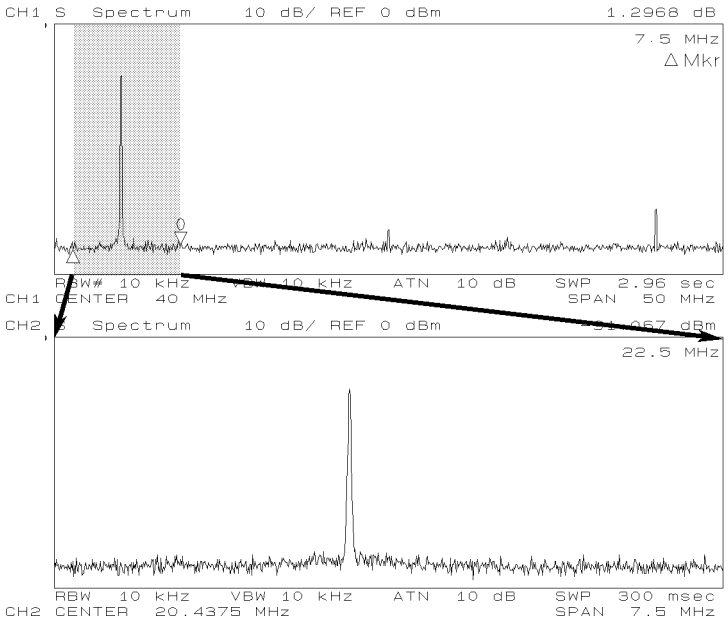


Figure 2-9. MKRΔ→SPAN

Step 5: Setting the Vertical Settings

You need to change the vertical settings if the top of peak is out of the grid or a signal level is too small relative to a reference setting. To change the vertical settings, change the reference value or spread the display area by changing the scale per division setting by doing one of the following procedures:

- To set the reference level
- To change the scale per division

To Set the Reference Level

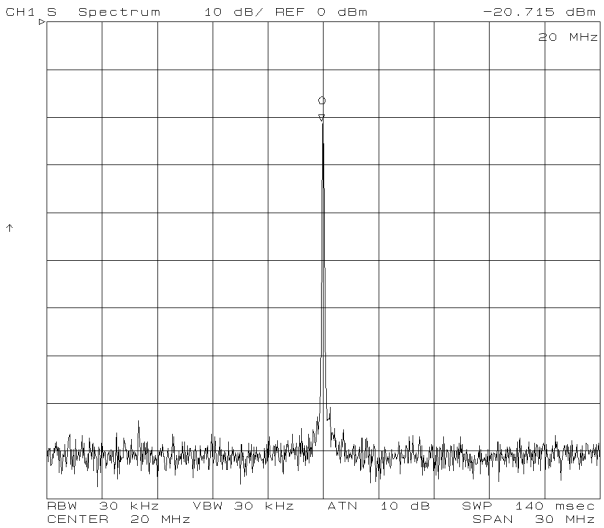
Using the Entry Keys

1. Press Scale Ref.
2. Press REFERENCE VALUE.

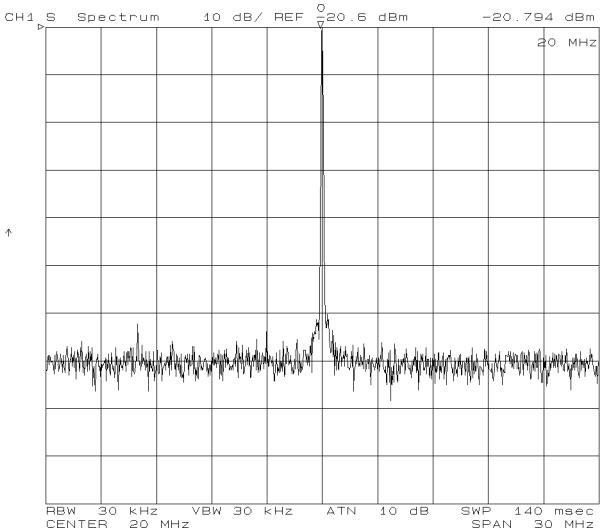
To	Use
Move trace toward top	or
Move trace toward bottom	or
Set reference value directly	0 . . . 9 and unit keys

Using the Marker

1. Press Search SEARCH:PEAK to move the marker to the peak.
2. Press Scale Ref.
3. Press MKR→REFERENCE.



Move the marker to the top of the peak



Press Scale Ref MKR→REFERENCE

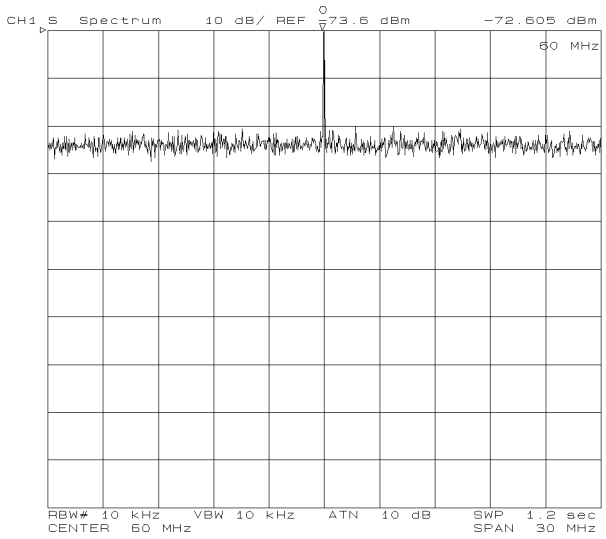
Figure 2-10. Marker to Reference

To Change the Scale per Division

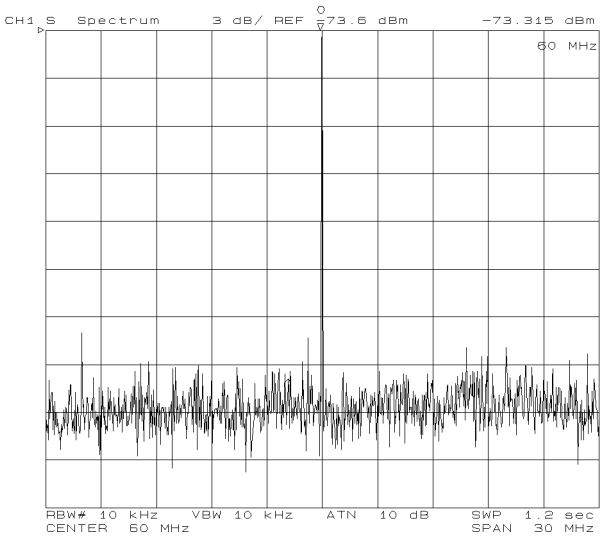
1. Set the reference level to the peak level of the target signal. See the "To Set the Reference Level" procedure.
2. Press (Scale Ref).
3. Press SCALE/DIV.
4. Change the scale/division setting to display additional details by using following keys:

To	Use
Change continuously	
Change 1-2-5 steps	
Set Scale/Div directly	(0) ... (9) and unit keys

This function can be used to display a small peak on a full grid.



Scale 10 dB/Div



Scale 3 dB/Div

Figure 2-11. Changing Scale/Div.

Step 6: Tuning the Settings

This step provides the bandwidth settings. The RBW setting affects the resolution of the frequency and lowers the displayed noise floor. The video bandwidth setting reduces noise variation.

- To set the resolution bandwidth (RBW)
- To set the video bandwidth
- To minimize the sweep time

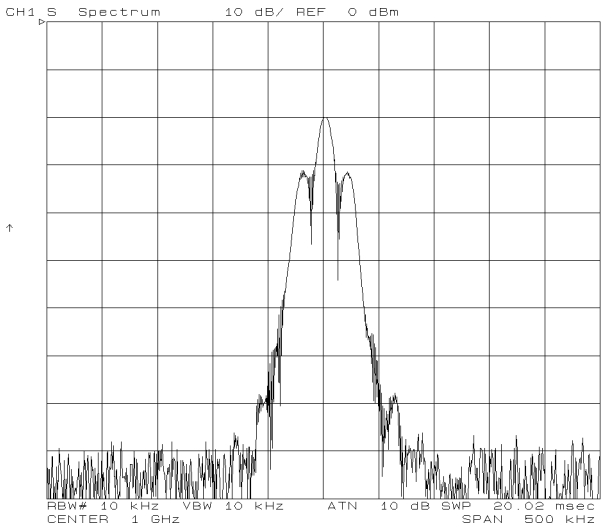
To Set the Resolution Bandwidth (RBW)

1. Press Bw/Avg.
2. Change the RBW setting by using ⇧, ⇩, or the knob.

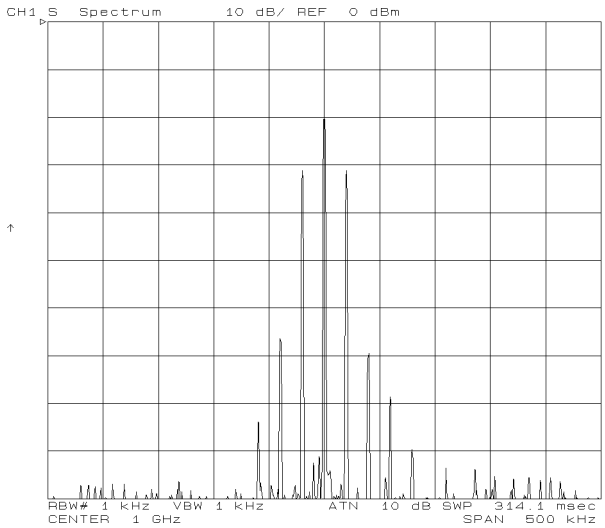
If the internal IF filter is wider than the difference of the adjacent signals, the analyzer cannot separate them. You must set the Resolution Bandwidth (RBW) narrower to make sure the analyzer can recognize each signal. For more information about RBW, see the *Function Reference*.

Narrowing the RBW reduces the noise power per display point. As a result, the displayed noise floor is down and the lower level signal is displayed.

For example, the trace of a 20 kHz amplitude modulated signal conceals sidebands in the skirt of the carrier trace when the RBW is 10 kHz. If you set the RBW to 1 kHz, the carrier and sidebands are split completely and the displayed noise floor is down.



RBW 30 kHz



RBW 1 kHz

Figure 2-12. Setting Resolution Bandwidth (RBW)

To Set the Video Bandwidth

1. Press Bw/Avg.
2. Press VIDEO BW.
3. Set video bandwidth by using the following keys:

To	Use
Lower noise level	↓, or 
Faster sweep time	↑, or 
Set bandwidth directly	0 . . . 9 and unit keys

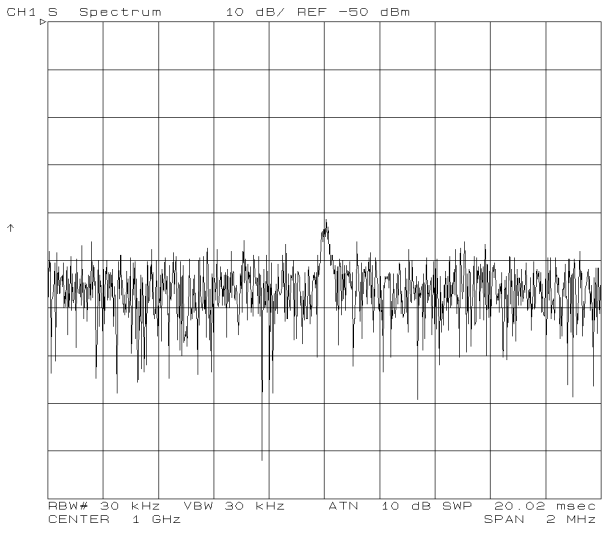
When the target signal and the noise are hard to distinguish because of noise variation, narrow the video bandwidth. This reduces the noise variations and makes the signal clearly visible. However, if the video bandwidth is narrowed, the sweep takes more time.

The allowable VBW setting is 1/1, 1/3, 1/10, 1/30, 1/100, and 1/300 of the current RBW setting.

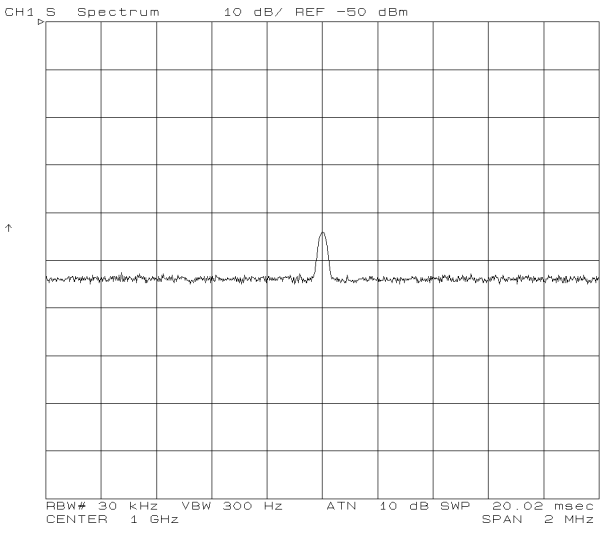
To Turn Off the Video Bandwidth

1. Press Bw/Avg VIDEO BW.
2. Enter the same value as the RBW setting.

For more information about the VBW, see chapter 12 of the *Function Reference* manual.



Video BW 10 kHz



Video BW 300 Hz

Figure 2-13. Setting Video Bandwidth (VBW)

To Minimize the Sweep Time

1. Press Sweep.
2. Press SWEEP TIME.
3. Press 0 ×1.

The analyzer reduces the NOP (number of points) to equalize the display resolution and the RBW. As a result, the displayed trace becomes rough. However, the frequency readout resolution does not change.

This feature is effective when the RBW is greater than Span/(NOP−1). For more information about this feature, see chapter 6 of the *Function Reference* manual.

Figure 2-14 shows an example of minimizing the sweep time. Center, span, VBW, and RBW settings are the same, but the sweep time that is shown on the right bottom of each trace is different. The sweep time of channel 2 is approximately 25% of channel 1.

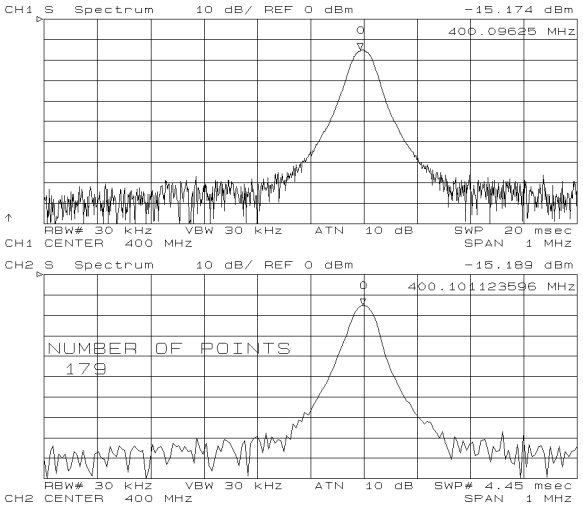


Figure 2-14. Minimized Sweep Time

Step 7: Performing Calibration

This step provides procedures for the reference level calibration. Performing the reference level calibration improves the level accuracy.

- To perform the reference level calibration
- To use the external reference
- To turn off the calibration

To Perform the Reference Level Calibration

1. Wait at least 30 minutes after the analyzer is turned on. This warming up period is required to meet the analyzer's specifications.
2. Attach the N(m)-BNC(f) adapter (furnished) to an input (S, R, A, or B). (See Figure 2-15.)
3. Connect the CAL OUT output and the selected input using the BNC cable (furnished).
4. Press (Meas). Then select the input (that the cable is connected to) by pressing `S`, `R`, `A` or `B`.
5. Press (Cal).
6. Press `EXECUTE LVL CAL`.

The analyzer automatically changes the frequency setting to measure the CAL OUT signal. When the reference level calibration is completed, all measurement settings are restored.

After the calibration is completed, "`Cor`" appears on the left side of the grid.

To obtain a higher level measurement accuracy, perform the reference level calibration just before reading the measured value. If you change the setting of the RBW or the attenuator, you must perform the reference level calibration again. The analyzer stores the calibration data individually for each of the inputs.

The error terms that are canceled by the reference level calibration are described in chapter 12 of the *Function Reference* manual.

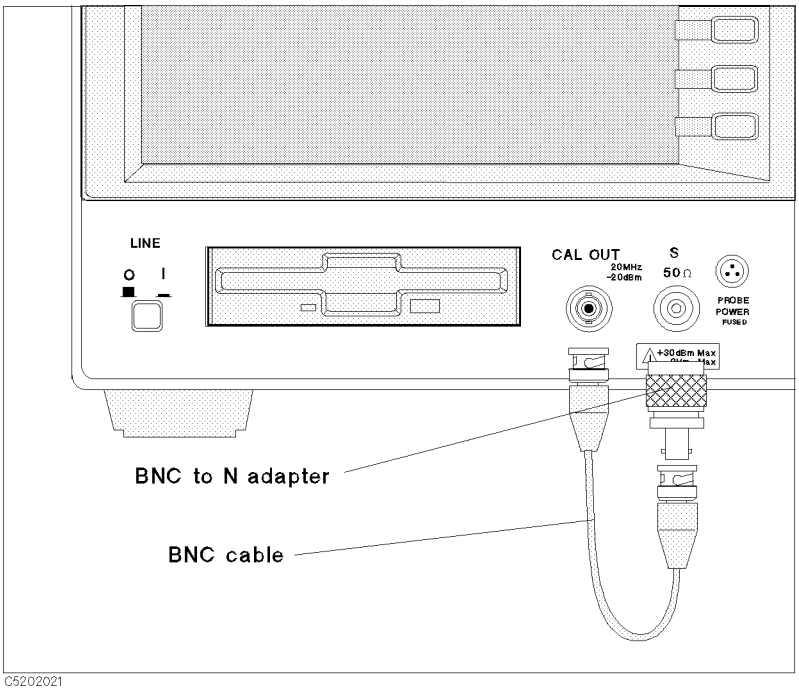


Figure 2-15. Reference Level Calibration Connection

To Use the External Reference

You can use an external reference signal instead of the internal reference.

1. Wait at least 30 minutes after the analyzer is turned on. This warming up period is required to meet the analyzer's specifications.
2. Connect the external reference to the selected front-panel spectrum input.
3. Set up the frequency range to measure the external reference signal.
4. If a unit other than dBm is selected, press (Format) dBm.
5. Press (Search) SEARCH:PEAK. Then move the marker to the peak of the external reference signal.
6. Read the external reference signal level.
7. Calculate the *expected level – readout level* of the reference signal.
8. Press (Cal) LVL CAL DATA. Then enter the calculated value.

To Turn Off Calibration

1. Press Cal
2. Press LVL CAL DATA. Then press 0 x1

The "Cor" notation disappears from the display.

To Perform the Calibration for the 75 Ω Configuration

If you want to perform a reference level calibration in the 75 Ω configuration:

1. If the analyzer is not set up for a 75 Ω configuration, see appendix A of the *User's Guide*.
2. Confirm that the analyzer is configured to measure the 75 Ω configuration by pressing Cal INPUT Z.
3. Wait at least 30 minutes after the analyzer is turned on. This warming up period is required to meet the analyzer's specifications.
4. Attach the 75 Ω N(m) to 50 Ω BNC(f) adapter to the 11852B option C04 50 Ω(m)/75 Ω(f) minimum loss pad that is connected to the S input. (See Figure 2-16.)
5. Connect the CAL OUT and the S input with the 50 Ω BNC cable.
6. Press EXECUTE LVL CAL.
7. After the completion of calibration, disconnect the BNC cable.
8. Detach the 75 Ω N(m) to 50 Ω BNC(f) adapter.

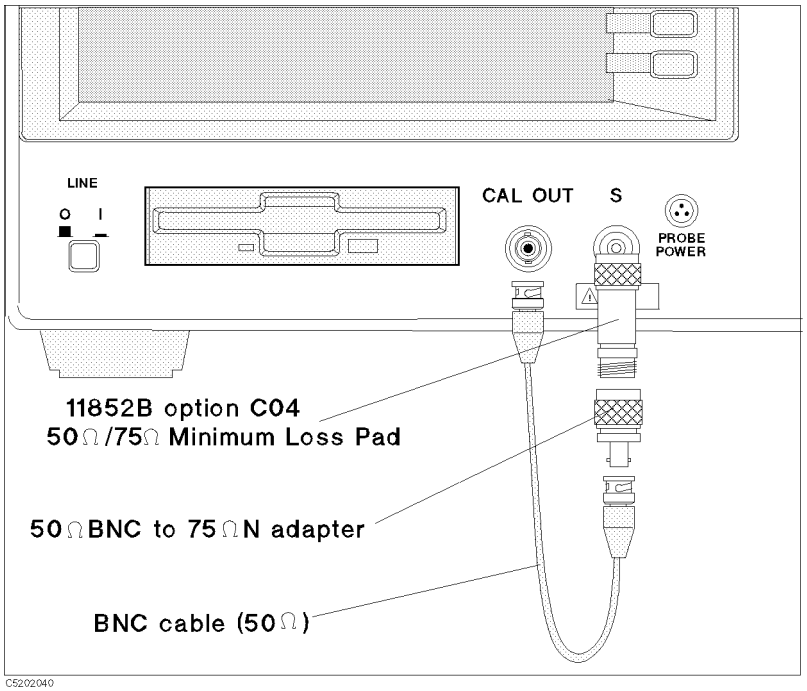


Figure 2-16.
Reference Level Calibration Connection for the 75 Ω Configuration

Step 8: Reading the Measured Result

If you can display the correct trace on the display, you can read the measured signal level using the marker. The analyzer has several useful search functions. This step provides the procedures for reading the measured results using the marker.

- To read a value using the marker
- To stabilize the trace
- To use the sub-markers
- To use the Δmarker
- To search for a single peak on the trace
- To search for multiple peaks
- To ignore small peaks

To Read a Value Using the Marker

1. Press (Marker).
2. Move the marker by performing the following steps:
   - Turn the rotary knob until the marker moves to the point where you want to read the measured value.
   - Enter the target frequency by using numerical keys.
3. Read the marker value displayed on the upper right of the display.

If you want a more accurate frequency reading of the target signal, set the span and the RBW as narrow as possible.

Note

The readout resolution of the frequency is determined by the setting of the frequency span, the number of points (NOP), and the resolution bandwidth (RBW). The resolution is the larger value between SPAN/(NOP−1) and RBW. For example, when the frequency span is 10 MHz, the NOP is 801, and the RBW is 10 kHz, the readout resolution is approximately 12.5 kHz.

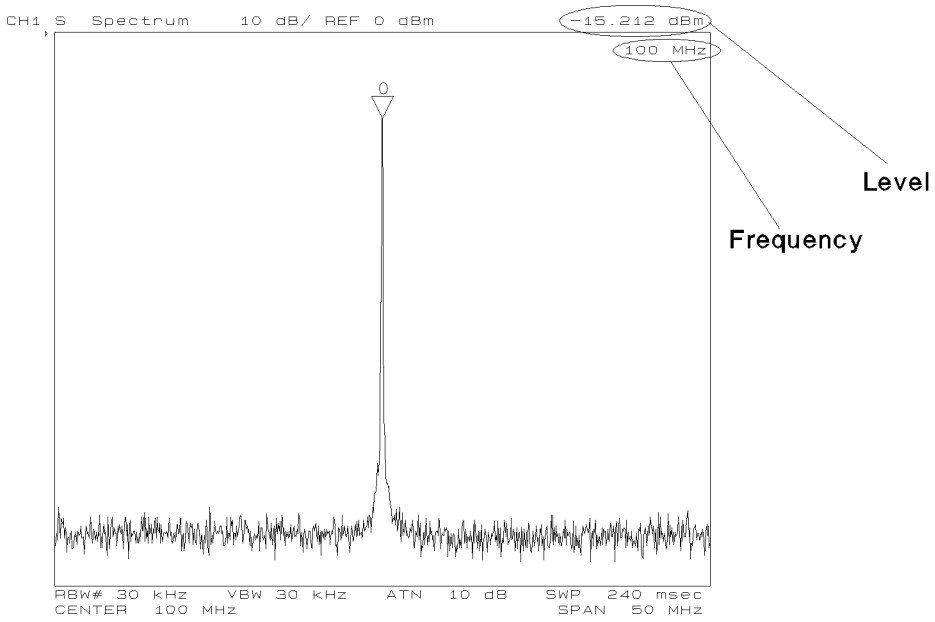


Figure 2-17. Marker Readout

To Stabilize the Trace

When the trace is not stable and the marker value changes frequently, it is difficult to read the measured value. You can use the following techniques to stabilize the trace:

- Stop the sweep.
- Use the averaging function.
- Use the maximum or minimum hold function.
- Capture the unstable signal using signal track.

To Stop the Sweep

1. Press (Trigger).
2. Press `SWEEP: HOLD`.

The sweep is stopped immediately (even if the sweep is in progress). If you want to restart the sweep, press `CONTINUOUS` to start a free-run sweep or press `SINGLE` to make a single sweep.

To Use the Averaging Function

1. Press (Bw/Avg).
2. Press `AVERAGING FACTOR`.
3. If needed, enter the averaging factor (number of times). Then press the (x1). Default averaging factor is 16.
4. Toggle `AVERAGING on OFF` to `ON off`.

The averaging notation (`Avg`) appears on left side of the grid when averaging is turned on. The averaging notation indicates the number of times averaging has been performed. When averaging is completed, the counter stops incrementing. However, the trace continues updating with each sweep.

Averaging requires a sweep with a specified number of times that is enough for an averaging factor to complete the averaging. You can set the number of sweeps by using the number of groups function.

If you want to change the setting of any parameter when averaging, you can restart averaging from the 0 count.

To restart the averaging, press (Bw/Avg) `AVERAGING RESTART`. This resets averaging counter to 0.

To Use Maximum or Minimum Hold Function

1. Press (Display).
2. Press `DATA HOLD [OFF]`.

To Hold	Press
Maximum Level	`MAX`
Minimum Level	`MIN`

"Max" (or "Min") appears on the right of the grid when the maximum (minimum) hold function is activated.

To turn off the maximum or minimum hold, press (Display) DATA HOLD [MAX] HOLD: OFF.

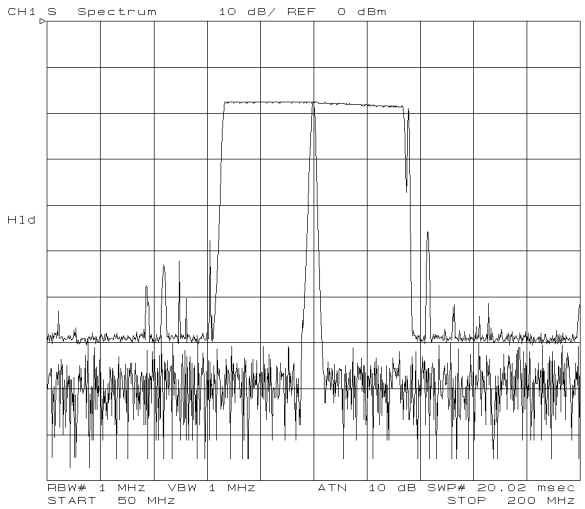


Figure 2-18. Maximum Holding the Drifting Signal

To Capture an Unstable Signal Using Signal Track

1. Press (Search).
2. Press SEARCH:PEAK to move the marker to the peak of the drifting test signal.
3. Toggle SIGNAL TRK on OFF to ON off.

The signal track function captures the peak that is indicated by the marker and places it in the center of the grid for each sweep. If the peak is unstable horizontally, use this function. The analyzer automatically changes the center frequency to keep the peak in the center of the grid.

To Use the Sub-markers

1. Press (Marker).
2. Move the marker to the point where you want to set the sub-marker.
3. Press SUB MKR.
4. Select the sub-marker from SUB MKR 1 to 7.
5. Press (Utility).
6. Toggle MKR LIST on OFF to ON off to display the marker list on the bottom of the display.

The sub-marker appears at the point of that the marker is displayed. Sub-markers are fixed horizontally and you cannot move them.

The sub-marker value can only be displayed by using the marker list.

To clear a sub-marker, press (Marker) CLEAR SUB MKR. Then press the sub-marker number that you want to erase from the display.

To clear all of the markers, press (Marker) PRESET MKRS.

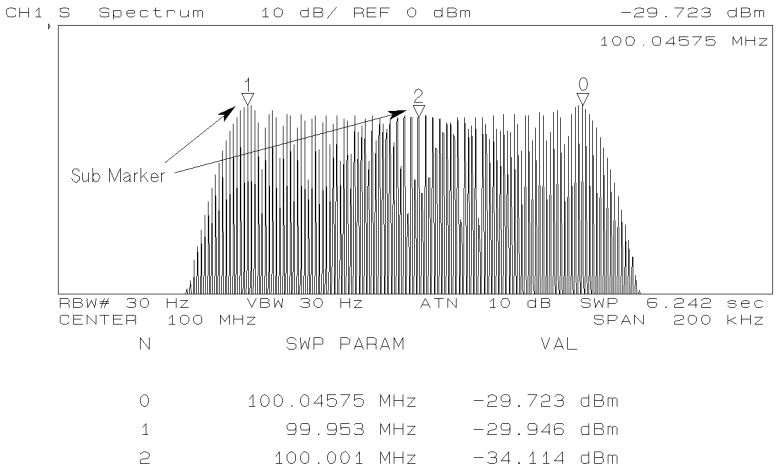


Figure 2-19. Sub-marker and Maker List

To Use the ΔMarker

1. Press Marker.
2. Place the marker at the point you want use as the reference point by using the  .
3. Press ΔMODE MENU.
4. Press ΔMKR.
5. The reference marker appears at the marker point.
6. To move the marker:
   - Enter an offset frequency by using the numerical keys.
   - Turn the rotary knob until the marker moves to the point you want to read the value.
7. Read the level and the frequency differences from the reference marker that are displayed on the upper right of the grid.

The marker value on the upper right of the grid shows the frequency and the level differences between the reference marker and the marker.

When you use the sub-markers, use the marker list to display the difference between reference the marker and the sub-markers.

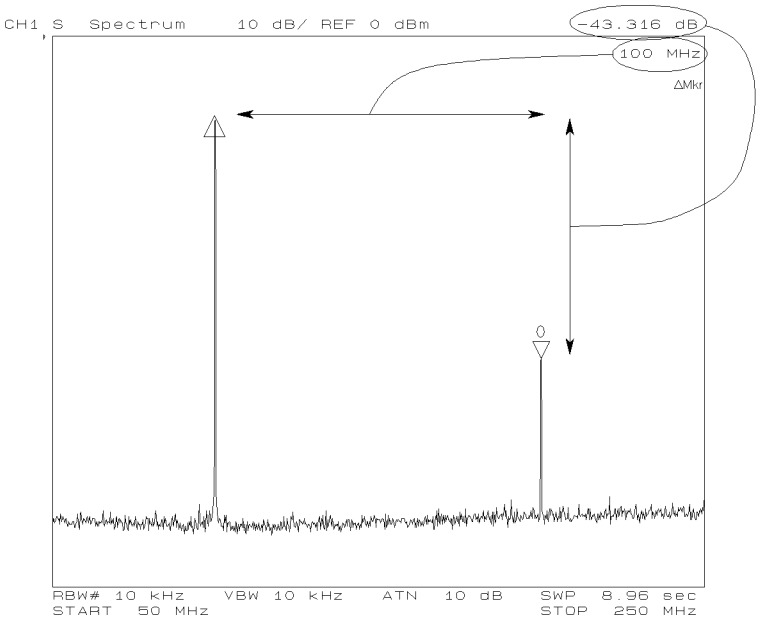


Figure 2-20. ΔMarker

To Search For a Single Peak on the Trace

1. Press Search.
2. Press SEARCH:PEAK to search a maximum peak.
3. If you want to search for another peak:

To search next peak for	Press
2nd highest peak	NEXT PEAK
Peak just to the left	NEXT PEAK LEFT
Peak just to the right	NEXT PEAK RIGHT

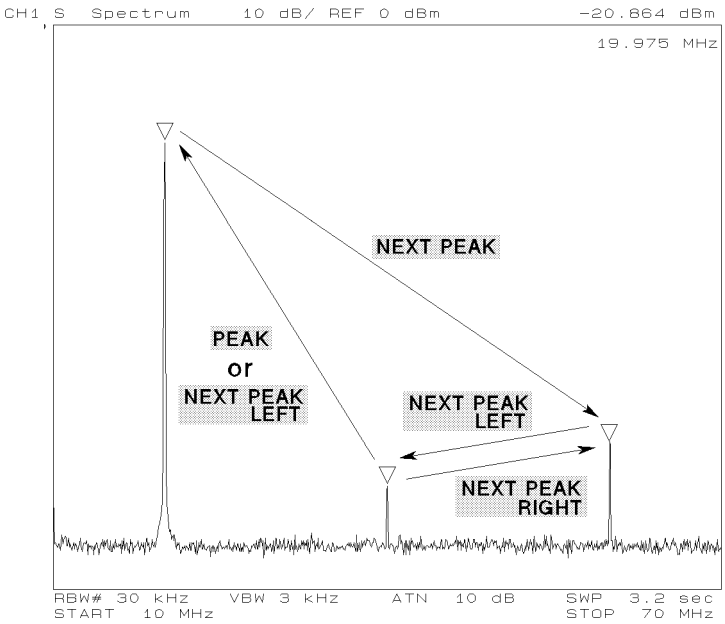


Figure 2-21. Peak Search

To Search For Multiple Peaks

1. Press (Search) MULTIPLE PEAKS.
2. Do any of following:

To search for peaks	Press
For the all peaks	SEARCH: PEAKS ALL
For peaks on the right	PEAKS RIGHT
For peaks on the left	PEAKS LEFT

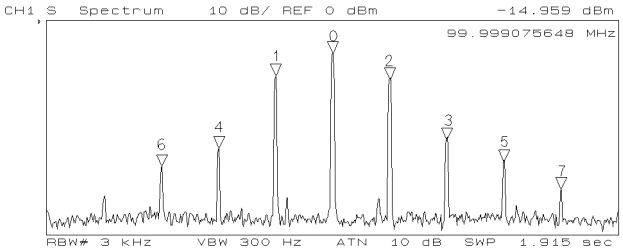
3. Press (Utility). Toggle MKR LIST on OFF to ON off to list all marker values.

When this function is enabled, the marker is placed on the maximum peak and the sub-markers are placed on up to seven other peaks. PEAKS ALL searches for all the peaks and places the sub-markers in the order of peak level.

PEAKS RIGHT and PEAKS LEFT search only to the right or left side of the peak and place the sub-markers on peaks in the order found.

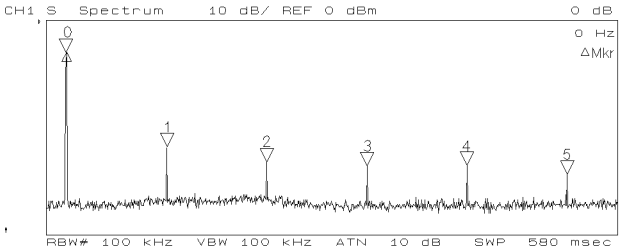
If the frequency of the carrier is unstable, use the search track function by toggling (Search) SEARCH TRK on OFF to ON off. The analyzer searches for the peaks each sweep to capture the shifted carrier and any harmonics or sidebands.

If the marker is to search for peaks other than harmonics, specify the peak threshold for the search function. This makes the search function ignore the peaks that have a lower level than the threshold level. See the "To Ignore Small Peaks" procedure.



N	SWP PARAM	VAL
0	99.999075648 MHz	-14.959 dBm
1	99.899075648 MHz	-26.508 dBm
2	100.099075648 MHz	-26.67 dBm
3	100.199075648 MHz	-54.55 dBm
4	99.800325648 MHz	-59.704 dBm
5	100.297825648 MHz	-65.179 dBm
6	99.700325648 MHz	-68.484 dBm
7	100.397825648 MHz	-78.953 dBm

PEAKS:ALL



N	SWP PARAM	VAL
0	0 Hz	0 dB
1	99.75 MHz	-43.599 dB
2	199.5 MHz	-51.512 dB
3	299.9625 MHz	-53.796 dB
4	399.7125 MHz	-51.908 dB
5	499.4625 MHz	-57.11 dB
Δ	99.95 MHz	-15.104 dBm

PEAKS RIGHT with Δmarker

Figure 2-22. Searching for Multiple Peaks

To Specify the Search Range

You can set the search function to search within a specified range. To specify the search range, use one of the following two procedures:

- Using the marker
- Using the Δmarker

Using the Marker

1. Press (Search).
2. Press SEARCH RANGE MENU.
3. Toggle PART SRCH on OFF to ON off.
4. Move the marker to the start point of the search range.
5. Press MKR→LEFT RNG to set the marker position as the left edge of the range.
6. Move the marker to the end point of the search range.
7. Press MKR→RIGHT RNG to set the marker position to the right edge of the range.
8. Press RETURN

Using the ΔMarker

1. Press (Marker).
2. Move the marker to the start point of the search range.
3. Press ΔMODE MENU.
4. Press ΔMKR.
5. Move the marker to the end point of the search range.
6. Press (Search).
7. Press SEARCH RANGE MENU.
8. Toggle PART SRCH on OFF to ON off to enable the search range.
9. Press MKRΔ→SEARCH RNG.

All the search functions search within a specified search range. You can specify the search range for each channel individually. The triangle-shaped indicator at the bottom of the grid shows the current search range (see Figure 2-23). In this figure, PEAK searches for the highest peak within the specified range. It does not search all of the grid.

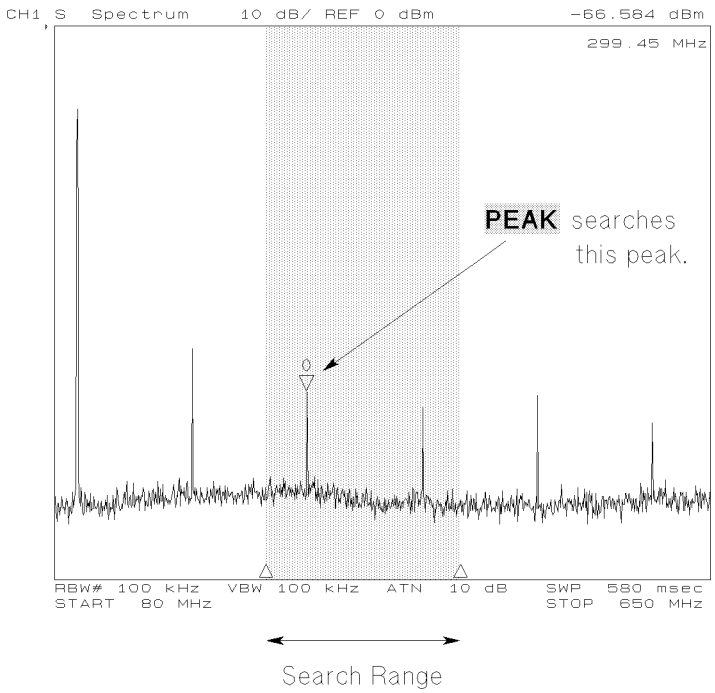


Figure 2-23. Search Range

To turn off the part search, toggle `PART SRCH ON off` to `on OFF`.

To Ignore Small Peaks

You can set the search function to ignore small peaks (such as noise) by defining the peak. The analyzer accept the following two types of peak definition.

- Peak threshold
- Peak height

Defining the Peak Threshold

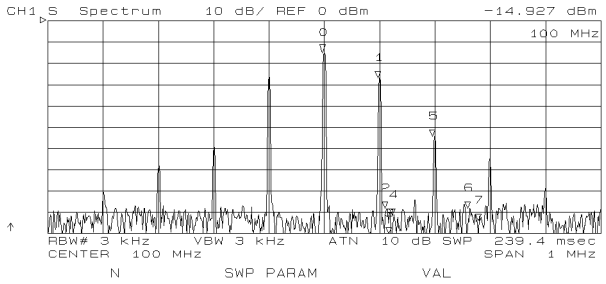
Using Rotary Knob.

1. Press (Search).
2. Press SEARCH:PEAK PEAK DEF MENU.
3. Toggle THRESHOLD on OFF to ON off to display the red threshold line.
4. Press THRESHOLD VALUE. Then set the threshold value using the following:
   - Enter the threshold value using numerical keys.
   - Turn the rotary knob to move the threshold line to the appropriate position.
5. Press RETURN

Using the Marker.

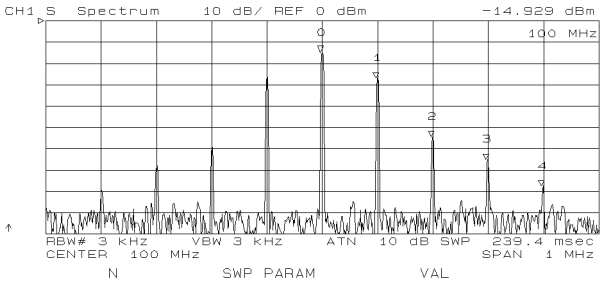
1. Press (Search) SEARCH:PEAK.
2. Move the marker to the point you want to set as a threshold value.
3. Press PEAK DEF MENU MKR→THRESHOLD.
4. Toggle THRESHOLD on OFF to ON off.
5. Press RETURN

When the peak threshold is activated, the red threshold line is displayed. The marker search function searches only for peaks above the threshold line.



N	SWP PARAM	VAL
0	100 MHz	-14.927 dBm
1	100.1 MHz	-26.663 dBm
2	100.1125 MHz	-88.291 dBm
3	100.12 MHz	-102.9 dBm
4	100.12625 MHz	-91.205 dBm
5	100.19875 MHz	-54.369 dBm
6	100.2625 MHz	-88.342 dBm
7	100.28125 MHz	-93.883 dBm

Before Defining the Threshold



N	SWP PARAM	VAL
0	100 MHz	-14.929 dBm
1	100.1 MHz	-26.659 dBm
2	100.2 MHz	-54.384 dBm
3	100.29875 MHz	-64.949 dBm
4	100.39875 MHz	-77.542 dBm

After Defining the Threshold

Figure 2-24. Threshold Function

Defining Peak Height

1. Press Search.
2. Press `SEARCH:PEAK` `PEAK DEF MENU`.
3. Press `PEAK DEF: ΔY`.
4. Enter a peak height using the numerical keys and the units terminator keys.
5. Press `RETURN`.

For more information about peak definition, see chapter 12 of the *Function Reference* manual.

3

Typical Spectrum Measurement Techniques

This chapter describes typical measurement techniques using the spectrum analyzer mode of operation.

The measurement techniques described in this chapter are as follows:

- To measure the noise level
- To measure the carrier to noise ratio
- To perform the time gated spectrum analysis
- To measure zero span (time domain measurement)
- To track unstable harmonics using search track
- To track and zoom a signal

To Measure the Noise Level

1. Press (Format).
2. Press NOISE.
3. Press (Scale Ref). Then press (⇩) until the noise trace closes to the reference level.
4. Press (Bw/Avg). Then press VIDEO BW.
5. Press (⇩) to flatten the noise trace.
6. Press (Marker). Then turn the  and read the normalized noise level.

The marker readout unit becomes "dBm/Hz" and is normalized by the 1 Hz equivalent noise bandwidth (ENBW). To convert the ENBW, see the "To Convert to a Different Equivalent Noise Bandwidth" procedure.

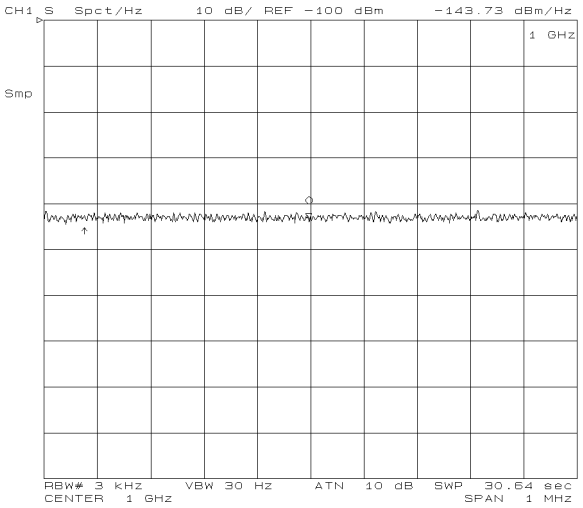


Figure 3-1. Noise Readout

Note

When you change the format to a noise format, the analyzer changes the detection mode to sample detection. If you return the format to SPECTRUM, the detection mode automatically is set to the positive peak detection mode. This happens even if you selected the negative detection mode before changing to the noise format.

To Convert to a Different Equivalent Noise Bandwidth

1. Calculate the conversion factor by using the following equations with displayed units:

Unit	Use
dBm/Hz	$K = 10 \log \frac{BW}{1}$
dBV/$\sqrt{Hz}$ and dBμV/$\sqrt{Hz}$	$K = 20 \log \frac{BW}{1}$
W/Hz	$K = 1/BW$
V/ $\sqrt{Hz}$	$K = 1/\sqrt{BW}$

   Where, *BW* is the different equivalent noise bandwidth.

2. Press (Display) DATA MATH.
   - Press OFFSET for dBm/Hz, dBV/$\sqrt{Hz}$, and dBμV/$\sqrt{Hz}$.
   - Press GAIN for V/$\sqrt{Hz}$ and W/Hz.
3. Enter *K*, then press (×1).

Note

The 4396B displays $dBV/\sqrt{Hz}$, $dB\mu V/\sqrt{Hz}$, $V/\sqrt{Hz}$ as dBV/Hz, dBμV/Hz, V/Hz respectively.

To Measure the Carrier to Noise Ratio

1. Set up the frequency range to measure a carrier signal.
2. Press (Search) SEARCH:PEAK to place the marker on the carrier signal.
3. Press (Scale Ref) MKR→REFERENCE to set the reference level to the carrier signal level.
4. Adjust the scale/div to display the carrier and noise floor. Use (Scale Ref) SCALE/DIV.
5. Press (Marker) ΔMODE MENU ΔMKR to place the reference marker on the carrier signal.
6. Press (Bw/Avg). Then press VIDEO BW.
7. Enter an appropriate video bandwidth to reduce the variation.
8. Press (Marker). Then do either of the following:
   - Enter the offset frequency by using the numeric keys.
   - Move the marker into the noise level of the trace by using the rotary knob.
9. If you want to normalize the marker readout with the RBW filter, press (Utility) and toggle NOISE FORM on OFF to ON off.
10. Read the difference from the reference marker.

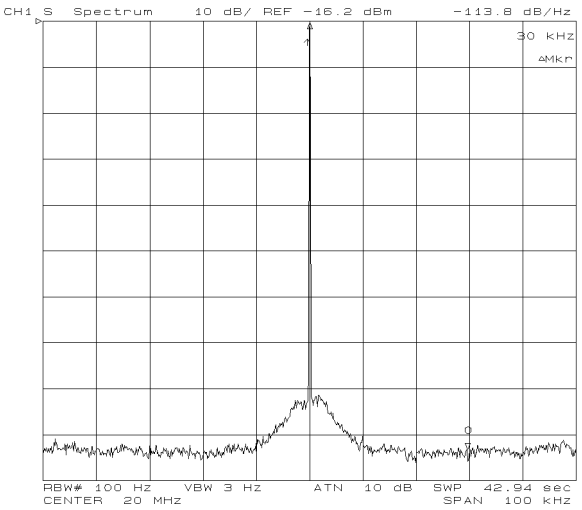


Figure 3-2. C/N Measurement

To Perform the Time Gated Spectrum Analysis

Do the following steps to perform a time gated spectrum analysis:

1. Determining Gate Trigger Parameters
2. Gate Trigger Source Connection
3. Setting Center and Span Frequency
4. Adjusting Gate Trigger
5. Setting RBW, VBW, and Averaging
6. Measuring

Note This function is only available with the option 1D6.

Step 1: Determining Gate Trigger Parameters

1. Connect the target signal and the trigger signal to an oscilloscope input (see Figure 3-3).

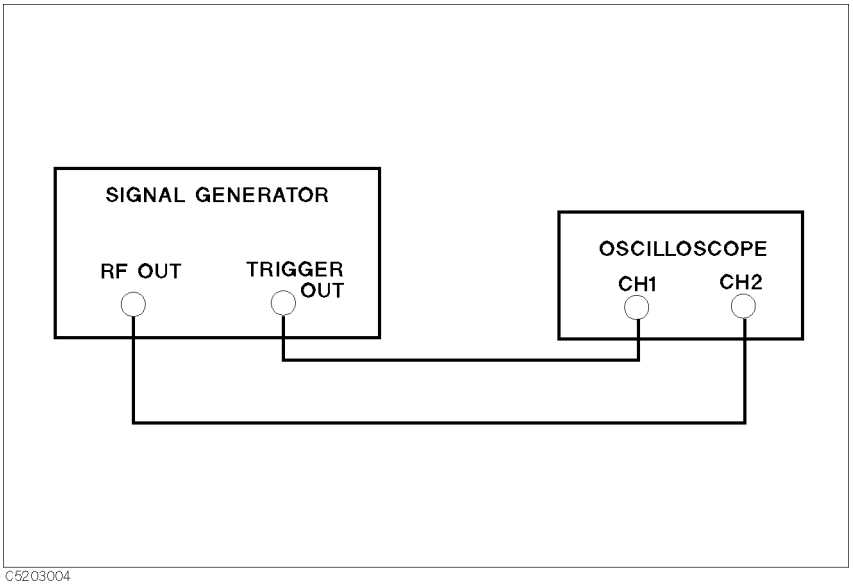


Figure 3-3. Time Domain Measurement Configuration

2. Adjust the oscilloscope to display the two signals.
3. Using the oscilloscope, check the following parameters:
   - For the target signal:
     - Signal width (τ)
     - Signal delay (SD)
   - For the trigger signal:
     - Pulse width (if you use the level trigger mode)

The signal delay (SD) is the delay inherent in the signal (that is, SD is the length of time after the trigger, but before the signal of interest occurred and becomes stable).

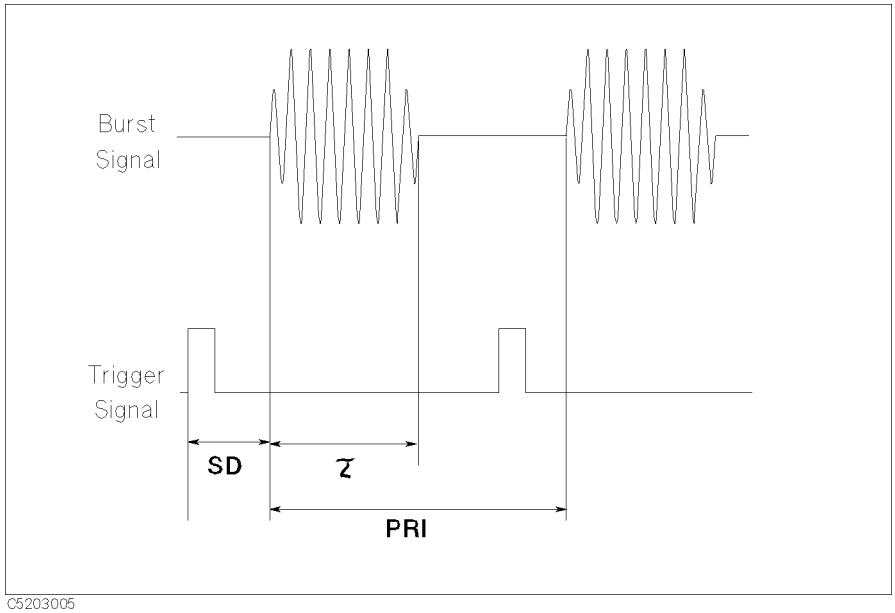


Figure 3-4. Target and Trigger Signal Timing on the Oscilloscope

4. Calculate the set up time (SUT) using, SUT = $\tau/2$.
5. Determine the gate parameters using the following equations:
   - Gate delay = SUT + SD
   - Gate length = $\tau/4$

Figure 3-5 shows the scheme of these parameters.

Open the "gate" during the time the signal is in a stable condition. The time from the start time of a signal and the open time of a gate is the "set up time" (SUT). Generally, SUT is set to half the pulse width. This is done so that you can allow the maximum time for the signal and RBW filter of the analyzer to become stable. You must also consider the end of the gate. Shut the gate before the last quarter of the pulse width to maintain a stable condition during the time the gate is open.

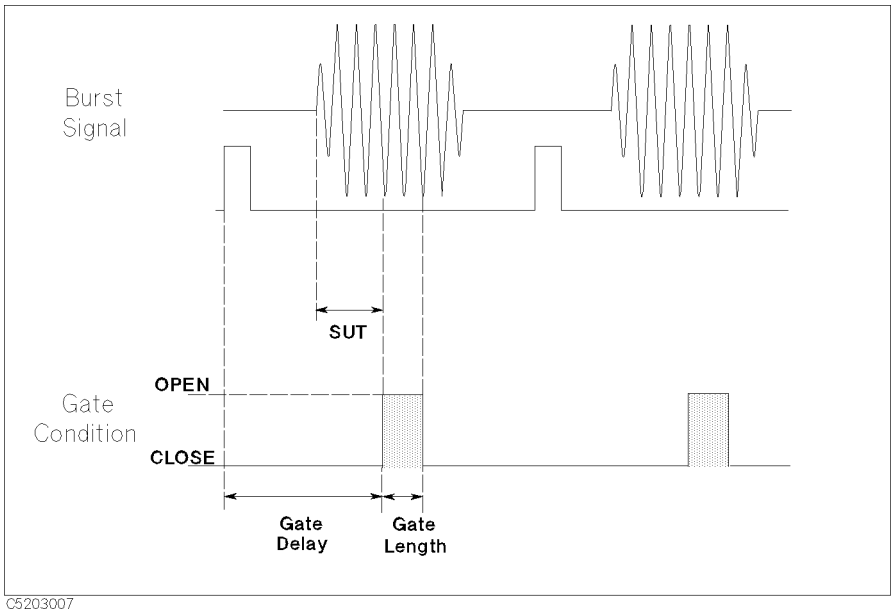


Figure 3-5. Gate Parameters

Step 2: Gate Trigger Source Connection

1. Connect the RF signal source to the S input of the analyzer.
2. Connect the trigger output from the signal source to the EXT TRIGGER connector on the rear panel of the analyzer.

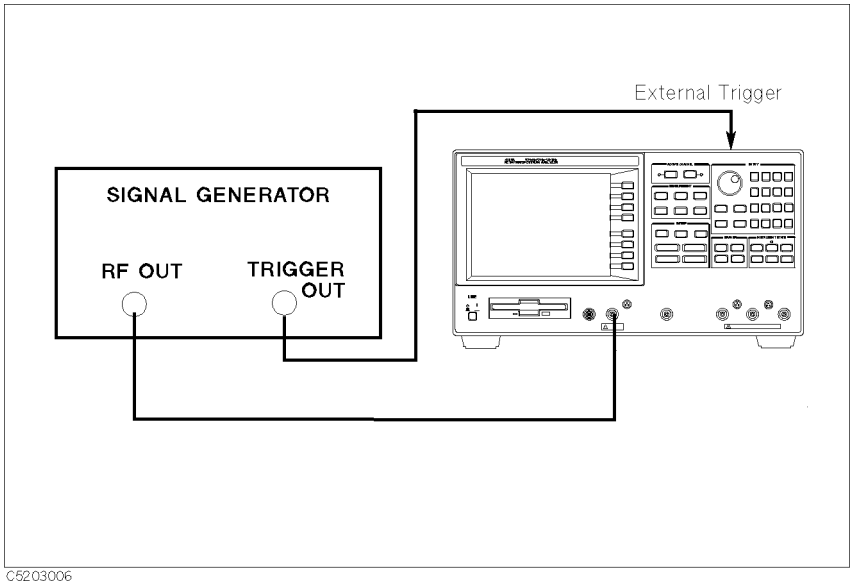


Figure 3-6. Time Gated Measurement Configuration

Step 3: Setting Center and Span Frequency

- Set up the center and span frequency of the analyzer to display the target signal.

Step 4: Adjusting Gate Trigger

1. Press (Trigger) CONTINUOUS to activate a gate trigger.
2. Press TRIGGER: [FREE RUN].
3. Press GATE [LEVEL].
4. Select the gate control mode. Select LEVEL or EDGE by toggling GATE CTL: LEVEL and EDGE to the required mode.
5. If you selected the LEVEL trigger mode, set the trigger polarity for starting the gate.
6. Press GATE DELAY.
7. Set a gate delay time.
8. If you select the EDGE trigger mode, press GATE LENGTH. Then set the gate open length.

You can see the gate trigger condition by monitoring the GATE Output terminal using an oscilloscope. The GATE Output terminal is located on the analyzer's rear panel.

For detailed information about the EDGE and LEVEL gate control modes, see the *Function Reference*.

Step 5: Setting RBW, VBW, and Averaging

To Set the Resolution Bandwidth

1. Press (Bw/Avg).
2. Set the resolution bandwidth wider than $\frac{2}{SUT}$.

The RBW setting you can use is limited by the gate open position. You can adjust a longer gate delay for a narrower RBW in step 5. This is because the RBW filters need charge time until normal condition. Therefore, you need to consider the balanced point between SUT and RBW. Generally, the RBW filter's charge time is defined as 2/RBW. Therefore, the SUT must be longer than 2/RBW or the RBW must be wider than 2/SUT.

Note

If the RBW is set narrower than 3 kHz, the analyzer turns into the stepped FFT mode. In the stepped FFT mode, the gate length must be longer than one FFT sampling step time. You must set the gate length longer than the minimum gate length that is listed in Table 3-1.

Table 3-1.
Minimum Gate Length setting with RBW setting

RBW	Allowable Gate Length
1 Hz	≥5.1857 s
3 Hz	≥1.6595 s
10 Hz	≥0.51858 s
30 Hz	≥0.13457 s
100 Hz	≥0.051213 s
300 Hz	≥0.012813 s
1 kHz	≥0.0032125 s
3 kHz	≥0.0016125 s

To Set the VBW

1. Press VIDEO BW.
2. Set the video bandwidth.

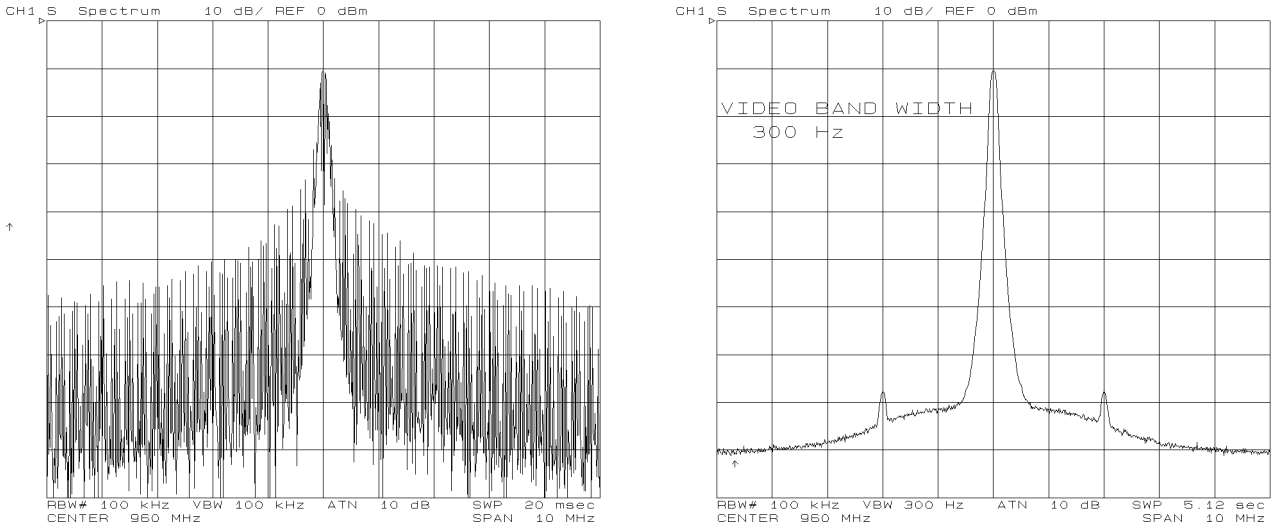
You can set any video bandwidth (VBW) without concern for the gate length setting. The analyzer implements the video filter using digital processing. The video filter of the analyzer requires no settling time for normal operation. Therefore, it is not affected by the gate length setting.

You can also use the averaging function to reduce the variation of the trace. The averaging function converges to the top of the variation. This is different from the VBW, which converges to the middle of the variation.

Step 6: Measuring

1. Adjust the span setting to fit the trace to your requirement.
2. Perform your measurement.

To Perform the Time Gated Spectrum Analysis



Before Time Gated

After Time Gated

Figure 3-7. Time Gated Spectrum Analysis

To Measure Zero Span

1. Determine the following parameters:
   - Sweep Time
   - Display Number of Points (NOP)
2. Press (Center). Then enter the frequency of the target signal.
3. Press (Span) `ZERO SPAN` to set the frequency span to 0 Hz.
4. Press (Sweep).
5. Select the sampling mode:
   - If (Sweep Time) / (NOP−1) $\geq$ 25 μs, then toggle to `SAMPLING [NORM rept]`. And Do both of step 6 and 7.
   - If (Sweep Time) / (NOP−1) $<$ 25 μs, then toggle to `SAMPLING [norm REPT]`. And do either step 6 or 7.
6. Set the sweep time, press (Sweep) `SWEEP TIME` then enter the sweep time.
7. Set the number of points (NOP), by pressing (Sweep) `NUMBER of POINTS` and entering the NOP.

The allowable minimum sweep time is determined by the sampling mode and NOP. In the normal sampling mode, the minimum sweep time is 25 μs $\times$ NOP.

In the repetitive sampling mode, the sweep time is determined by 0.5 μs $\times$ NOP. This is because that the time resolution is fixed at 0.5 μs in the repetitive sampling mode.

Notes

You can set the RBW greater than 10 kHz. If you enter less than 10 kHz for the RBW, it is set to 10 kHz.

You must set the trigger source either `EXTERNAL` or `VIDEO` to select the repetitive sampling mode.

The detection mode is automatically set to the sample detection mode.

To Stabilize the Trace Using the Video Trigger

1. Press (Trigger).
2. Press `TRIGGER: [FREE RUN]`.
3. Press `VIDEO`.
4. Adjust the video trigger level by using  to trigger the sweep at appropriate trace level.

If the sweep time and the pulse repetition rate (PRI) are not synchronized, the trace does not appear in the same position on the grid on every sweep. To avoid this, change the video trigger setting as appropriate.

To Read a Time Transition Using the Marker

1. Press (Utility).
2. Toggle `MKR TIME on OFF` to `ON off`.
3. Move the marker by using the  .
4. Read the transition time that is displayed on the upper right of the grid.

When in zero span measurement, the marker displays the same frequency on every point of trace. Using the marker time function, you can change the marker display to the time format instead of the frequency. The marker displays a time transition from the left end of the grid.

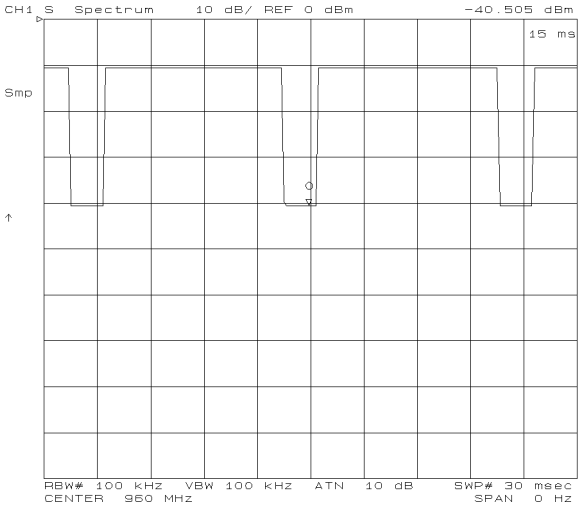


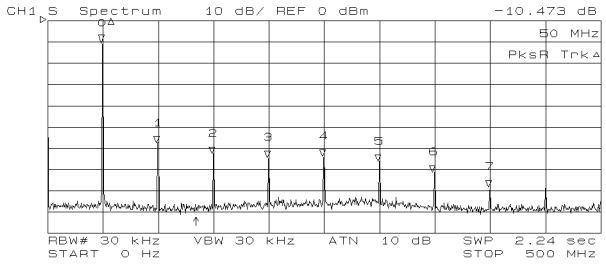
Figure 3-8. Marker Time

To Track Unstable Harmonics Using Search Track

1. Set the frequency range to display the carrier and the harmonics.
2. Press (Search) `SEARCH:PEAK` to move the marker to the peak.
3. Press (Marker) `ΔMODE MENU` `TRACKING ΔMKR` to set up the marker as a reference Δmarker that can move with the carrier.
4. Toggle (Search) `SEARCH TRK on OFF` to `ON off` to enable the search function on every sweep.
5. Press `MULTIPLE PEAKS` `SEARCH:PEAKS RIGHT` to search for carrier and harmonics under the search track.
6. Press (Utility). Then toggle `MKR LIST on OFF` to `ON off`.

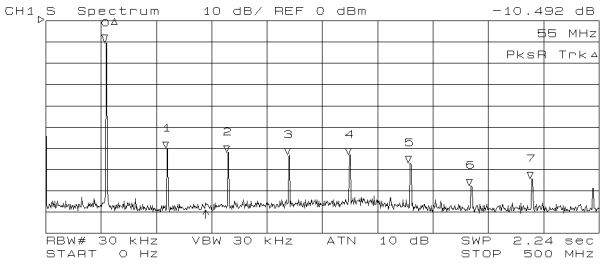
Even if the frequency of a carrier changes, the analyzer automatically tracks the carrier and the harmonics on the end of the sweep. Then the analyzer lists the difference between the carrier and the harmonics on the lower display.

If necessary, use the peak threshold to ignore the peaks other than the harmonics. See "To Ignore Small Peaks" in Chapter 2.



N	SWP PARAM	VAL
1	50 MHz	-46.978 dB
2	100 MHz	-52.01 dB
3	150 MHz	-54.185 dB
4	200 MHz	-53.585 dB
5	250 MHz	-55.747 dB
6	299.375 MHz	-61.059 dB
7	350 MHz	-67.855 dB
Δ	50 MHz	-10.473 dBm

Carrier Frequency: 50 MHz



N	SWP PARAM	VAL
1	55 MHz	-49.298 dB
2	110 MHz	-51.081 dB
3	165 MHz	-52.848 dB
4	220 MHz	-52.463 dB
5	274.375 MHz	-56.63 dB
6	329.375 MHz	-67.221 dB
7	384.375 MHz	-63.639 dB
Δ	55 MHz	-10.492 dBm

55 MHz

Figure 3-9. Tracking Unstable Harmonics Using Search Track

To Track and Zoom a Signal

Step 1: Setting the Wide Frequency Range

1. Set both channels to the spectrum analyzer mode.
2. Press (Display). Then toggle `DUAL CHAN on OFF` to `ON off`.
3. Select either channel to track the signal in wide span. Then set a frequency range that can display the signal as you change the frequency.
4. Press (Marker→).
5. Toggle `CROSS CHAN on OFF` to `ON off`
6. Press (Search) `PEAK`.
7. Toggle `SIGNAL TRK on OFF` to `ON off`.

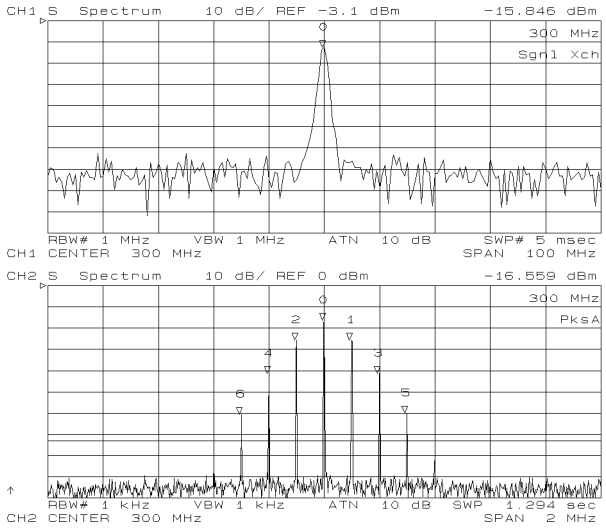
Step 2: Setting the Narrow Frequency Range for the Other Channel

1. Select the other channel to display the zoomed signal in a narrow span.
2. Press (Span).
3. Enter the frequency span to display the detail of the target signal. Note that the span of the zoomed channel must be wider than the frequency resolution × 2 of the channel that is set in step 1.
4. Press (Search). Then toggle `SEARCH TRK on OFF` to `ON off`.
5. Use `PEAK` (or the marker search functions) to analyze the target signal.

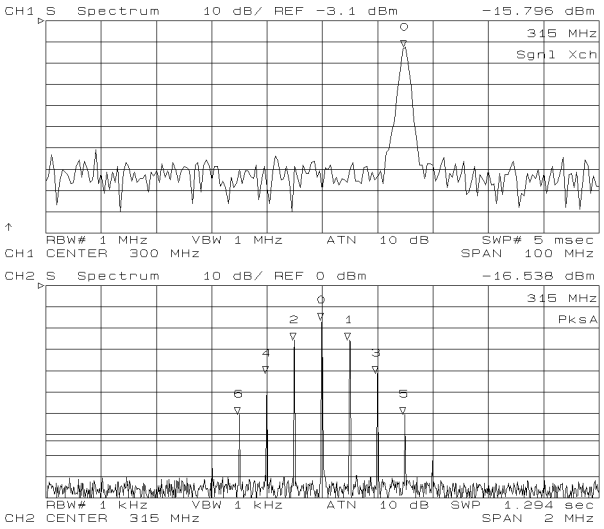
The analyzer displays a wide view on the wide span channel and a detailed view on the zoomed channel. When the frequency of the signal is changed, the wide span channel tracks the signal and the center frequency of the zoomed channel automatically changes to display the target signal.

When the marker list is turned on, wide span channel disappears from the display. However, the wide span channel keeps tracking the signal in the background and displays the results on the zoomed channel.

For more information about frequency resolution, see "To Read a Value Using the Marker" in Chapter 2.



300 MHz Carrier



315 MHz Carrier

Figure 3-10. Tracking and Zooming a Signal

Note

If the span of the zoomed channel is wider than the RBW of the wide channel, you can reduce the sweep time of the wide span channel. If the sweep time is reduced, the analyzer can track the faster movement of the target signal.

1. Press (Ch 1) or (Ch 2) to activate the wide span channel that has the active signal track function.
2. Press (Bw/Avg). Then set the RBW to the maximum setting that is narrower than the span of the zoomed channel.
3. Press (Sweep). Then press SWEEP TIME.
4. Press (0) (X1).

4

Recommended Network Measurement Task Sequence

This chapter describes a typical task sequence for any measurement using the network analyzer mode.

If you are using the analyzer for the first time, see the *User's Guide* first. The *User's Guide* provides the information needed to install and set up the analyzer, and a quick start guide to introduce you to the analyzer.

The measurement tasks described in this chapter are as follows:

- Step 1: Preparing for a measurement
- Step 2: Setting the sweep
- Step 3: Selecting the measurement format
- Step 4: Setting the sweep parameter
- Step 5: Setting the vertical settings
- Step 6: Tuning the settings
- Step 7: Performing calibration
- Step 8: Reading the measured result

Generally, you can make a measurement by performing these steps. If you want to perform a more complex network measurement, chapter 5 provides additional measurement techniques.

Step 1: Preparing for a Measurement

This step provides the procedures needed to prepare for a network measurement. You must perform the following tasks before you select measurement parameters (such as frequency range):

- To connect the test device
- To preset the analyzer
- To select the active channel
- To select the network analyzer mode

To Connect the Test Device

You must setup the test set before performing the network measurement. See appendix A of the *User's Guide* for the set up information of the test sets.

- Connect the device under test (DUT) as shown in Figure 4-1 or Figure 4-2.

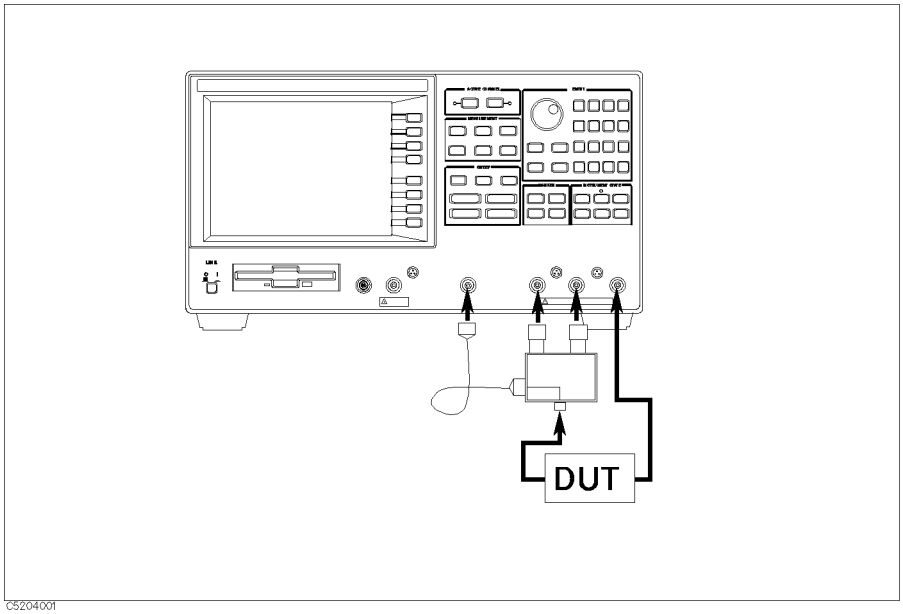


Figure 4-1. Using the Transmission/Reflection Test Set

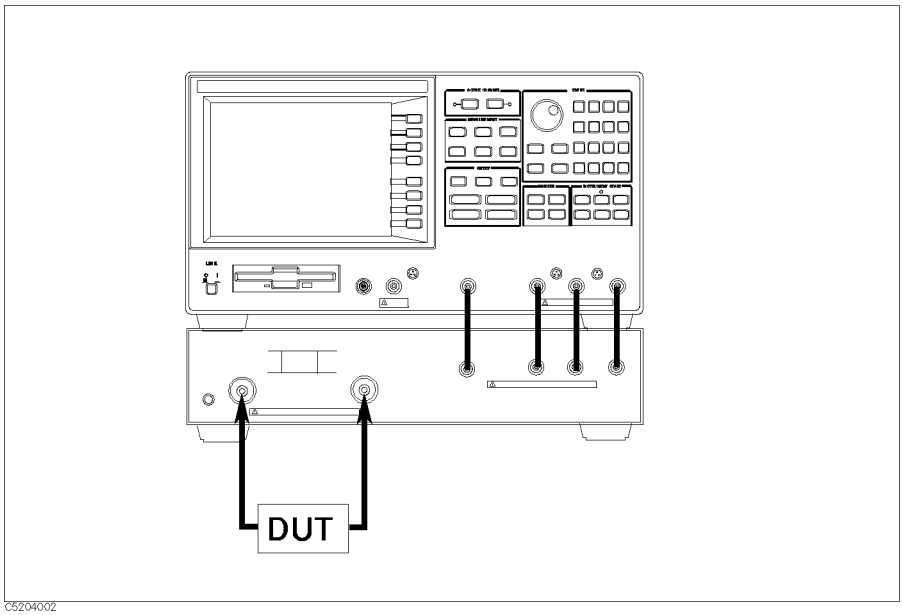


Figure 4-2. Using the S-Parameter Test Set

To Preset the Analyzer

- In the INSTRUMENT STATE block, press the green Preset key to set the analyzer to the preset state.

For additional information about the preset state, see the *Function Reference*.

To Select the Active Channel

- In the ACTIVE CHANNEL block, press Ch 1 (channel 1) or Ch 2 (channel 2) to select the active channel.

The analyzer has two independent channels. Each channel can have a different setting that includes whether the mode of operation is network analyzer or spectrum analyzer. Select the active channel before you set up any other settings.

Using this feature, you can change immediately to either of different two settings by pressing a single key.

To Select the Network Analyzer Mode

1. Press Meas.
2. Press `ANALYZER TYPE` `NETWORK ANALYZER`.

When you change the analyzer type, the analyzer is set to the preset state. So, when you want to change the analyzer type, you must select it before you set up the other settings or just after you select the active channel.

Step 2: Setting the Sweep

This step selects the trigger source and the sweep condition.

- To select the trigger mode
  - To use the external trigger
  - To trigger on each display point
- To select the sweep condition
- To select the sweep type
  - To use the power sweep

To Select the Trigger Mode

1. Press (Trigger).
2. Press TRIGGER:[FREE RUN] .
3. Select the trigger mode:

To Select a Trigger to	Press
Free run	FREE RUN
External trigger source	EXTERNAL [1]
Manual	MANUAL

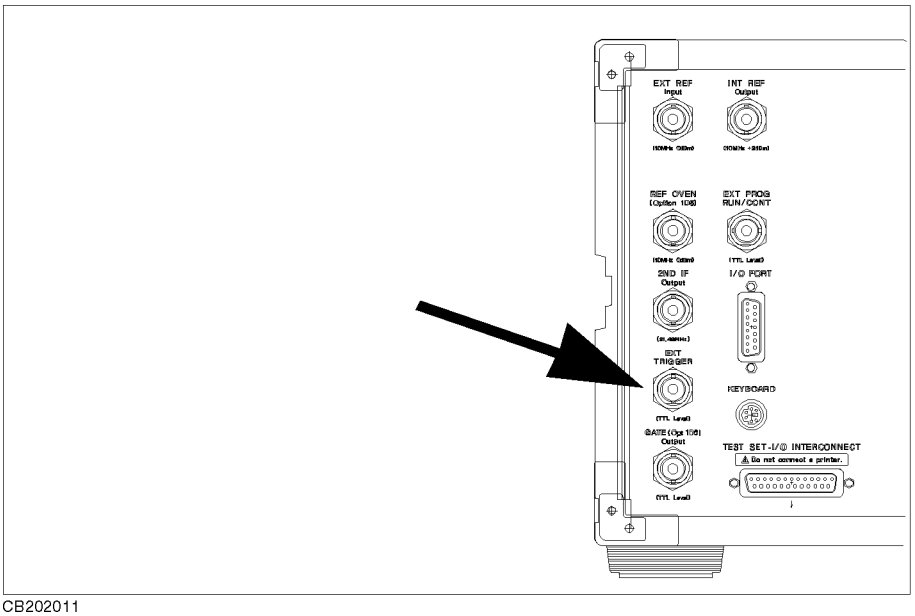
1 See the "To Use the External Trigger" procedure.

VIDEO and GATE (option 1D6 only) are only used for the spectrum measurements.

To Use the External Trigger

1. Connect the trigger source to the EXT TRIGGER connector on the rear panel of the analyzer.
2. Press (Trigger).
3. Press TRIGGER:[FREE RUN] .
4. Press EXTERNAL .
5. Input a trigger signal to the analyzer.

The external trigger signal level must be TTL Level.



CB202011

Figure 4-3. Location of EXT TRIGGER Connector

To Set the Trigger Signal Polarity.

1. Press Trigger.
2. Press `TRIGGER:[FREE RUN]`.
3. Toggle `TRIG PLRTY POS neg` to `pos NEG` to turn the trigger polarity to the negative logic.

To Trigger on Each Display Point

1. Press Trigger.
2. Press `TRIGGER: [FREE RUN]`.
3. Select a trigger source:

Trigger source	Do
Manual	Press `MANUAL`.
External	Press `EXTERNAL`.

4. Toggle `TRIG EVENT [ON SWEEP]` to `[ON POINT]`.
5. Apply a trigger event by pressing `MANUAL` (for `MANUAL`) or entering the external trigger signal (for `EXTERNAL`).

The sweep indicator ("↑") is moved to each point by the trigger event. You can select this mode only for `MANUAL`, `EXTERNAL`, or the bus trigger mode. For more information about the bus trigger mode, see the *GPIB Command Reference*.

To Select the Sweep Condition

1. Press Trigger.
2. Choose one of the following:

To Sweep	Do
Continuously	Press CONTINUOUS.
Single Time	Press SINGLE.
Specified Times	Press NUMBER of GROUPS. Then enter the number of times to sweep.

To Select the Sweep Type

1. Press Sweep SWEEP TYPE MENU.
2. Choose one of the following:

Sweep type	Do
Linear Frequency	Press LIN FREQ.
Log Frequency	Press LOG FREQ.
List Frequency	See "To Make a List Sweep" in Chapter 6.
Power Sweep	See the "To Use the Power Sweep" procedure.

To Use the Power Sweep

1. Press Source CW FREQ. Then enter the CW frequency.
2. Press Sweep.
3. Press SWEEP TYPE MENU.
4. Press POWER SWEEP.
5. Enter the start and stop power.

For example, to sweep from −20 dBm to 10 dBm, press Start 2 0 (−) (×1), Stop 1 0 (×1).

You can set the sweep power with 0.1 dB resolution.

Step 3: Selecting the Measurement Format

This step provides following procedures for selecting measurement format:

- To select the input port
- To select the measurement format
- To display trace in a smith chart
- To use the impedance conversion function

To Select the Input Port

With the T/R Test Set

- Press (Meas).

To measure	Type	Press
Reflection	Ratio	A/R
Transmission	Ratio	B/R
Reflection	Absolute	A
Transmission	Absolute	B
Source	Absolute	R

With the S-Parameter Test Set

1. Set up the S-parameter test set. See Appendix A of the *User's Guide*.
2. Press (Meas).

To measure	Direction	Press
Reflection	Forward	Refl:FWD S11 [A/R]
Transmission	Forward	Trans:FWD S21 [B/R]
Transmission	Reverse	Trans:REV S12 [B/R]
Reflection	Reverse	Refl:REV S22 [A/R]

To Select the Measurement Format

1. Press (Format).
2. Select a measurement format from the following:

Selecting the Measurement Format

To Select	Press
LOG Magnitude	LOG MAG
Phase	PHASE
Group Delay	DELAY [1]
Smith Chart	SMITH CHART [2]
Polar Chart	POLAR CHART
Liner Magnitude	LIN MAG
Standing Wave Ratio (SWR)	SWR
Real Part Only	MORE FORMAT: REAL
Imaginary Part Only	MORE IMAGINARY
Expanded Phase	MORE EXPANDED PHASE
Admittance Chart (Inv. Smith Chart)	MORE ADMITTANCE CHART

1 See "To Display the Group Delay" in Chapter 5.

2 See the "To Display Trace As a Smith Chart" procedure.

If you selected an absolute measurement by selecting R, A or B under the (Meas) key, only the following formats can be selected:

To Select	Press
LOG Magnitude	LOG MAG
Liner Magnitude	LIN MAG
Standing Wave Ratio (SWR)	SWR

For more information about the measurement format, refer to chapter 5 of the *Function Reference* manual.

To Display Trace As a Smith Chart

1. Press (Format).
2. Press SMITH CHART to display the smith chart.
3. Use the marker to read a measured value, by pressing (Marker) and rotating  .

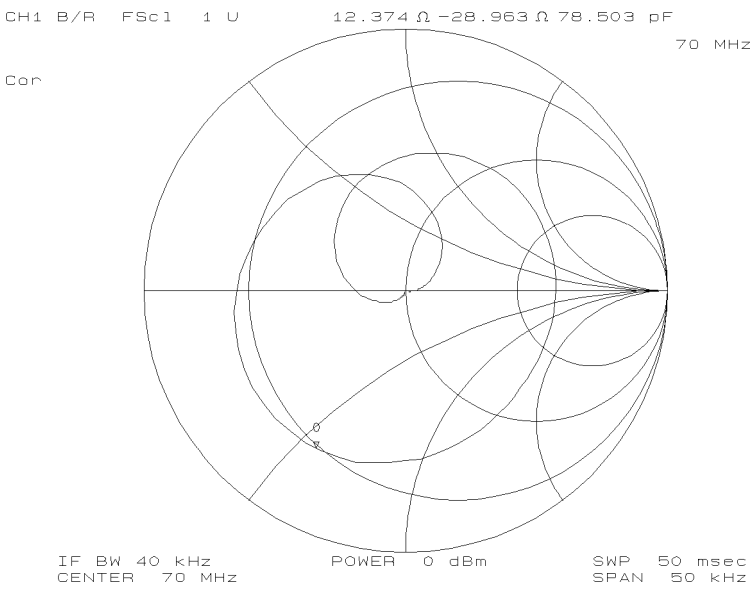


Figure 4-4. Smith Chart

To change the marker readout format, use the following procedure:

To Change Marker Readout Format

1. Press (Utility) SMTH/POLAR MENU.
2. Select display format using the following keys:

Format	Press
Real and Imaginary	CHART FMT: REAL IMAG
Linear Magnitude and Phase	LIN MAG PHASE
Log Magnitude and Phase	LOG MAG PHASE
Impedance	R+jX
Admittance	G+jB
SWR and Phase	SWR PHASE

To Use Impedance Conversion Function

1. Press (Meas).
2. Press CONVERSION [OFF].
3. Select one of the formats listed below:

Convert to	Selected Port	Press
Impedance	A/R, S11, or S12	`Z:Refl`
	B/R, S21, or S22	`Z:Trans`
Admittance	A/R, S11, or S12	`Y:Refl`
	B/R, S21, or S22	`Y:Trans`

The marker readout value is a linear impedance or admittance value even if the LOG MAG format is selected.

For additional information about the impedance conversion function, see the *Function Reference*.

Step 4: Setting the Sweep Parameter

To Set the Sweep Parameter Using [Start] and [Stop]

1. Press [Start] to activate the start frequency function.
2. Change the start frequency setting using the following keys:

To	Use
Set directly	[0] . . . [9] and units terminator keys
Change continuously	
Change with 1-2-5 steps	[↑] [↓]

3. Press [Stop].
4. Enter the stop frequency in a similar way.

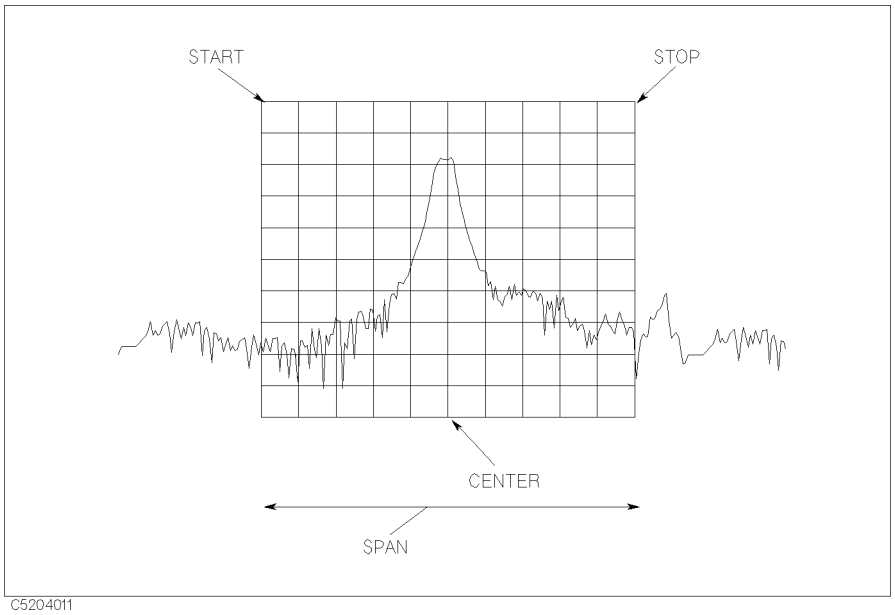


Figure 4-5. Sweep Parameter Setting

You can set the sweep parameter using [Center] and [Span] instead of [Start] and [Stop]. See "Step 4: Setting the Frequency Range" in Chapter 2.

Step 5: Setting the Vertical Settings

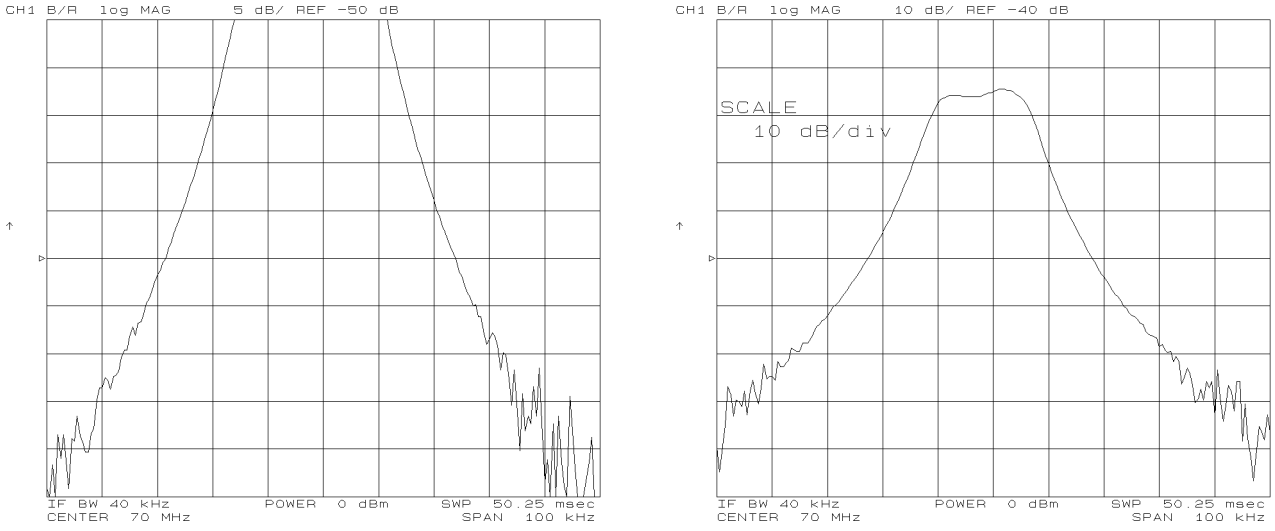
It is necessary to change a vertical setting if the trace is out of the grid or is too flat to see the required characteristics. This step provides the procedure needed to set the vertical scaling of the analyzer:

- To set the scale and reference automatically

To Set the Scale and Reference Automatically

1. Press Scale Ref.
2. Press AUTO SCALE to fit the trace within the grid.

The scale and reference are automatically adjusted to obtain an optimum trace display.



Before Auto Scale

After Auto Scale

Figure 4-6. Autoscale Function

Notes If you want to set the scale and reference value manually, the following functions are available:

- To change the scale per division setting, press SCALE/DIV.
- To change the reference position that is shown as "▶", use REFERENCE POSITION and ⇧ ⇩ keys.
- To change the reference value, use REFERENCE VALUE.

If you are displaying a data trace and a memory trace together, you need to consider whether you want to change the scale for one or both traces. You can change the traces as follows:

- If you want to change the scale setting for the data trace only, set `SCALE FOR [DATA]` and `D&M SCALE[UNCOUPLE]` under (Scale Ref) key.
- If you want to change the scale setting for the memory trace only, set `SCALE FOR [MEMORY]` and `D&M SCALE [UNCOUPLE]`.
- If you want to change the scale settings for the both traces, set `D&M SCALE [COUPLE]`.

Step 6: Tuning the Settings

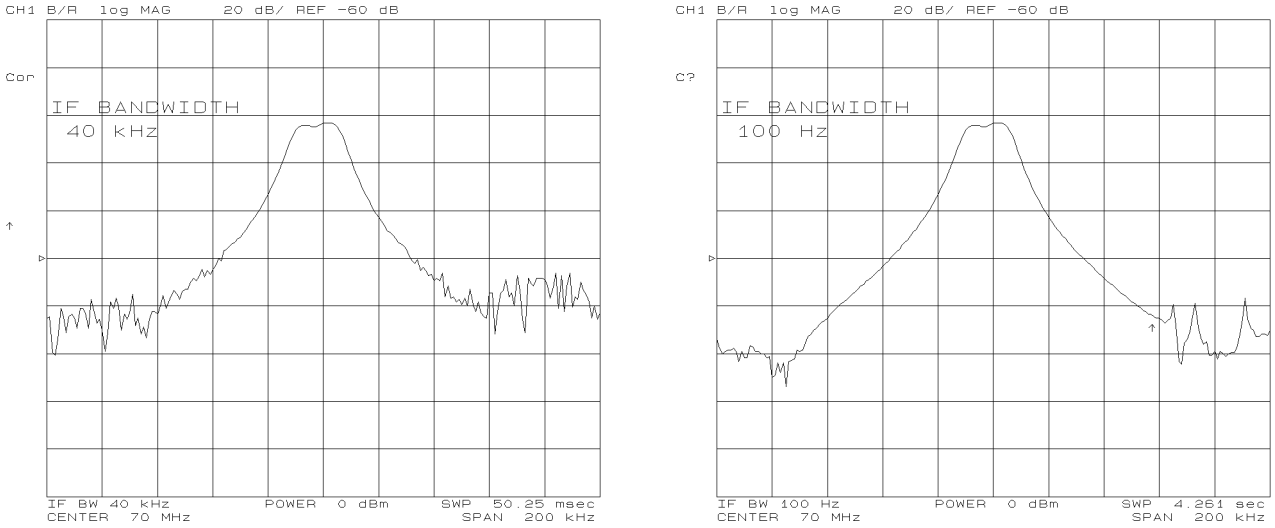
This step provides the following procedure:

- To increase the dynamic range

To Increase the Dynamic Range (Setting IF Bandwidth)

1. Press (Bw/Avg).
2. Press IF BW.
3. Press (↑) or (↓), or enter an IF bandwidth value directly from the numeric keypad.

Narrowing the IF bandwidth setting increases the dynamic range but it makes the sweep speed slower.



IF Bandwidth 40 kHz

IF Bandwidth 100 Hz

Figure 4-7. Setting IF Bandwidth (IFBW)

Notes

You can increase the dynamic range by using the following techniques:

- Apply the highest allowable power. The output power can be set by pressing (Source) POWER.
- Use the averaging. See "To Reduce Variation of the Measurement Trace Using Averaging".

Step 7: Performing Calibration

This step provide procedures for performing calibration. The calibration cancels the errors and improves the measurement accuracy. The analyzer has six different methods of calibration. You can select the method that fits your measurement requirement by reading "To Select an Appropriate Calibration Method" procedure. This step also contains a procedure to customize a calibration kit.

- To select an appropriate calibration method
- To perform a response calibration
- To perform a response & isolation calibration
- To perform an S11 1-port calibration
- To perform an S22 1-port calibration
- To perform a full 2-port calibration
- To perform a 1-path 2-port calibration
- To select the calibration kit
- To customize the user defined calibration kit

To Select an Appropriate Calibration Method

The analyzer has six calibration methods. You can choose the appropriate calibration method to fit your measurement by using Table 4-1.

Table 4-1. Calibration Method Selection Table

Measurement Type	Calibration Method	Complexity	See
Transmission or reflection measurement when the highest accuracy is not required.	Response	simple	"To Perform a Response Calibration"
Transmission of high insertion loss devices or reflection of high return loss devices. Not as accurate as 1-port or 2-port calibration.	Response & isolation	simple	"To Perform a Respornse & Isolation Calibration"
Reflection of any one-port device or well terminated two-port device.	S_{11} 1-port	slightly complex	"To Perform an S11 1-Port Calibration"
Reflection of any one-port device or well terminated two-port device.	S_{22} 1-port	slightly complex	"To Perform an S22 1-Port Calibration"
Transmission or reflection of highest accuracy for two-port devices. S-parameter Test Set is required.	Full 2-port	complex	"To Make Full 2-Port Calibration"
Transmission or reflection of highest accuracy for two-port devices. (Reverse test device between forward and reverse measurements.)	One-path 2-port	complex	"To Perform a 1-Path 2-Port Calibration"

To Perform a Response Calibration

1. Press (Cal) `CALIBRATE MENU` `RESPONSE` to display the response calibration menu.
2. Connect one of following standards. Then press the corresponding key.

Measurement Type	Connect Standard	Press
Transmission Measurement	THRU	`THRU`
Reflection Measurement	OPEN	`OPEN`
	SHORT	`SHORT`

3. Press `DONE: RESPONSE`.

To Perform a Response & Isolation Calibration

1. Press (Cal) `CALIBRATE MENU` `RESPONSE & ISOL'N` to display the response and isolation calibration menu.
2. Press `RESPONSE`.
3. See 2 of the "To Perform a Response Calibration" procedure.
4. Press `DONE: RESPONSE`.
5. Connect isolation standard (LOAD).
6. Press `ISOL'N STD`.
7. Press `DONE RESP ISOL'N CAL`.

To Perform an S11 1-Port Calibration

Step 1: Opening the S-11 1-Port Calibration Menu

1. Press (Cal).
2. Select the proper calibration kit. If the connector type or calibration kit name shown in the `CAL KIT` softkey label is not the same as the calibration kit to be used, see the "To Select the Calibration Kit" procedure.
3. Press `CALIBRATE MENU` `S11 1-PORT`.

Step 2: Measuring the OPEN

1. Connect OPEN standard to port 1.
2. Press `(S11):OPEN` (for the 7 mm or 3.5 mm cal kit) or `(S11):OPENS` (for the type-N cal kit).

   When the 7 mm or 3.5 mm calibration kit is selected, the message "`WAIT - MEASURING CAL STANDARD`" is displayed while the OPEN data is measured. The softkey label `OPEN` is then underlined. Skip to step 3.
3. If the type-N calibration kit is selected, do the following.
   a. Press `OPEN [M]` (for a male port connector) or press `OPEN [F]` (for a female port connector).

      The OPEN data is measured and the softkey label is then underlined.
   b. Press `DONE:OPENS`.

Step 3: Measuring the SHORT

1. Disconnect the OPEN. Then connect a SHORT standard to port 1.
2. Press `SHORT` (for the 7 mm or 3.5 mm calibration kit) or `SHORTS` (for the type-N calibration kit).

   When the 7 mm or 3.5 mm calibration kit is selected, the SHORT data is measured and the softkey label is underlined. Skip to step 4.
3. If the type-N calibration kit is selected, do the following.
   a. Press `SHORT [M]` (for a male port connector) or press `SHORT [F]` (for a female port connector).

      The SHORT data is measured and the softkey label is then underlined.
   b. Press `DONE:SHORTS`.

Step 4: Measuring the LOAD

1. Disconnect the SHORT, and connect an impedance-matched LOAD (usually 50 or 75 Ω) at port 1.

2. Press LOAD. Then wait the LOAD is measured and the LOAD softkey is underlined.

Step 5: Completing the Calibration

1. Press DONE 1-PORT CAL to complete the calibration.

The error coefficients are computed, and the correction menu is returned to the screen with CORRECTION ON. A corrected S11 trace is displayed, and "Cor" appears at the left side of the screen.

If you press DONE without measuring all the required standards, the message "CAUTION: ADDITIONAL STANDARDS NEEDED" is displayed.

To Perform an S22 1-Port Calibration

This calibration is similar to the S11 1-port calibration except that S22 is selected automatically. It is used only with an S-parameter test set.

For S-parameter measurements in the reverse direction with a transmission/reflection test kit, use the S11 1-port or 1-path 2-port calibration and reverse the DUT between measurement sweeps.

To Make Full 2-Port Calibration

Step 1: Calling the Full 2-Port Calibration Menu

1. Press (Cal).
2. Select the proper calibration kit. If the connector type or calibration kit name shown in the `CAL KIT` softkey label is not the same as the calibration kit to be used, see the "To Select the Calibration Kit" procedure.
3. Press `CALIBRATE MENU` `FULL 2-PORT` `REFLECT'N`.

Step 2: Measuring the Reflection

1. Connect a shielded OPEN to port 1.
2. Press `(S11): OPEN` (for the 7 mm or 3.5 mm calibration kit) or `(S11): OPENS` (for the type-N calibration kit).

   When the 7 mm or 3.5 mm calibration kit is selected in step 1, the OPEN data is measured and the softkey label `OPEN` is underlined.
3. If the type-N calibration kit is selected, do the following.
   a. Press `OPEN [M]` (for a male port connector) or press `OPEN [F]` (for a female port connector).

      The OPEN data is measured. The softkey label is then underlined.
   b. Press `DONE:OPENS`.
4. Disconnect the OPEN and connect the SHORT to port 1.
5. Press `(S11): SHORT` (for the 7 mm or 3.5 mm calibration kit) or `(S11): SHORTS` (for the type-N calibration kit).

   When the 7 mm or 3.5 mm calibration kit is selected, the SHORT data is measured and the softkey label `SHORT` is underlined.
6. If the type-N calibration kit is selected, do the following.
   a. Press `SHORT [M]` (for a male port connector) or press `SHORT [F]` (for a female port connector).

      The SHORT data is measured and the softkey label is then underlined.
   b. Press `DONE:SHORTS`
7. Disconnect the SHORT and connect an impedance-matched LOAD (usually 50 or 75 Ω) at port 1.
8. Press `(S11): LOAD`. Then confirm the `LOAD` softkey label is underlined.
9. Repeat the OPEN-SHORT-LOAD measurements described above, connecting the devices in turn to port 2 and using the (S22) softkeys.

10. Press REFLECT'N DONE.

The reflection calibration coefficients are computed and stored. The two-port calibration menu is displayed (with the REFLECT'N softkey underlined).

Step 3: Measuring the Transmission

1. Press TRANSMISSION.
2. Connect a THRU connection between port 1 and port 2 at the points where the test device is connected.
3. When the trace settles, press FWD. TRANS. THRU. Then wait S21 frequency response is measured and the softkey label is underlined.
4. Press FWD. MATCH THRU. Then wait the S11 load match is measured and the softkey label is underlined.
5. Press REV. TRANS. THRU. Then wait the S12 frequency response is measured and the softkey label is underlined.
6. Press REV. MATCH THRU. Then wait the S22 load match is measured and the softkey label is underlined.
7. Press TRANS. DONE.

The transmission coefficients are computed and stored. The two-port calibration menu is displayed (with the TRANSMISSION softkey underlined).

Step 4: Measuring the Isolation

1. If correction for isolation is not required, press ISOLATION OMIT ISOLATION ISOLATION DONE. Then skip to step 5.
2. If correction for isolation is required, connect impedance-matched LOADs to port 1 and port 2.
3. Press FWD ISOL'N ISOL'N STD. Then wait the S21 isolation is measured and the softkey label is underlined.
4. Press REV ISOL'N ISOL'N STD. Then wait the S12 isolation is measured and the softkey label is underlined.
5. Press ISOLATION DONE.

The isolation error coefficients are stored. The two-port calibration menu is displayed (with the ISOLATION softkey underlined).

Step 5: Completing the Calibration

1. Press DONE: 2-PORT CAL to complete the calibration.

The error coefficients are computed and stored. The correction menu is displayed (with CORRECTION ON). A corrected trace is displayed. The notation "C2" at the left of the screen indicates that two-port error correction is ON.

Now the test device can be connected and measured. Save the calibration data on the built-in disk drive.

To Perform a 1-Path 2-Port Calibration

Step 1: Calling the 1-Path 2-Port Calibration Menu

1. Press Cal.
2. Select the proper calibration kit. If the connector type or calibration kit name shown in the `CAL KIT` softkey label is not the same as the calibration kit to be used, see the "To Select the Calibration Kit" procedure.
3. Press `CALIBRATE MENU` `ONE-PATH 2-PORT` `REFLECT'N`.

Step 2: Measuring the Reflection

1. Connect a shielded OPEN to the test port.
2. Press `(S11): OPEN` (for the 7 mm calibration kit) or `(S11): OPENS` (for the type-N calibration kit).

   The OPEN data is measured, and the softkey label `OPEN` is underlined.
3. If the type-N calibration kit is selected, do the following.
   a. Press `OPEN [M]` (for a male port connector) or press `OPEN [F]` (for a female port connector).

      The OPEN data is measured and the softkey label is then underlined.
   b. Press `DONE:OPENS`.
4. Disconnect the OPEN and connect a SHORT to the test port.
5. Press `SHORT` (for the 7 mm or 3.5 mm calibration kit) or `SHORTS` (for the type-N calibration kit).

   When the 7 mm or 3.5 mm calibration kit is selected, the SHORT data is measured and the softkey label `SHORT` is underlined.
6. If the type-N calibration kit is selected, do the following.
   a. Press `SHORT [M]` (for a male port connector) or press `SHORT [F]` (for a female port connector).

      The SHORT data is measured and the softkey label is then underlined.
   b. Press `DONE:SHORTS`.
7. Disconnect the SHORT and connect an impedance-matched LOAD (50 or 75 Ω) to the test port.
8. Press `LOAD`. Then wait the LOAD is measured and the softkey label `LOAD` is underlined.
9. Press `REFLECT'N DONE`.

The reflection calibration coefficients are computed and stored. The two-port calibration menu is displayed (with the REFLECT'N softkey underlined).

Step 3: Measuring the Transmission

1. Connect a THRU between the test port and the return cable to the analyzer (connect to the points at which the test device is connected).
2. Press TRANSMISSION.
3. Press FWD. TRANS. THRU. Then wait the S21 frequency response is measured and the softkey label is underlined.
4. Press FWD. MATCH THRU. Then wait the S11 load match is measured and the softkey label is underlined.
5. Press TRANS. DONE.

The transmission coefficients are computed and stored. The two-port calibration menu is displayed (with the TRANSMISSION softkey underlined).

Step 4: Measuring the Isolation

1. If correction for isolation is not required, press ISOLATION OMIT ISOLATION ISOLATION DONE. Skip to step 5.
2. If correction for isolation is required, connect impedance-matched LOADs to the test port and the return port.
3. Press FWD ISOL'N ISOL'N STD. Then wait the S21 isolation is measured and the softkey label is underlined.
4. Press ISOLATION DONE.

The isolation error coefficients are stored. The two-port calibration menu is displayed (with the ISOLATION softkey underlined).

Step 5: Completing the Calibration

1. Press DONE 2-PORT CAL to complete the calibration.

The error coefficients are computed and stored. The correction menu is displayed with CORRECTION ON. A corrected trace is displayed. The notation "C2" at the left of the screen indicates that 2-port error correction is ON.

Step 6: Performing the Measurement

1. Connect the test device in the reverse direction. Then press PRESS to CONTINUE.
2. Reconnect the test device in the forward direction. Then press PRESS to CONTINUE.

Now the error corrected trace is displayed. If you measure the other test device, press (Trigger) MEASUREMENT RESTART. Then perform the procedure of step 6.

Save the calibration data on a disk or RAM disk.

For additional information about calibration, see chapter 5 of the *Function Reference*.

To Select the Calibration Kit

1. Press Cal.
2. Press CAL KIT [...].
3. Press one of the following keys:

Calibration Kit	Press
7 mm calibration kit	CAL KIT: 7mm
3.5 mm calibration kit	3.5mm
50 Ω N type	N 50Ω
75 Ω N type	N 75Ω
User defined calibration kit	USER KIT

To Customize the User Defined Calibration Kit

To Define the Standard Definition

Step 1: Preparation.

1. Prepare the "*Standard Definitions*" table of the standard kit you want to use.

Table 4-2 is an example of a standard definition table.

Table 4-2.
Example of the Standard Definitions (85032B 50Ω Type-N Calibration Kit)

STANDARD		C0	C1	C2	C3	FIXED OR	OFFSET			FREQUENCY (GHz)		COAX or	STANDARD
NO.	TYPE	$\times10^{-15}$F	$\times10^{-27}$F	$\times10^{-36}$F/Hz	$\times10^{-45}$F/Hz	SLIDING	DELAY ps	LOSS MΩ/s	Z_0 Ω	MINIMUM	MAXIMUM	WAVEGUIDE	LABEL
1	SHORT						0	700	50	0	999	COAX	SHORT(M)
2	OPEN	108	55	130	0		0	700	50	0	999	COAX	OPEN(M)
3	LOAD					FIXED	0	700	50	0	999	COAX	BROAD-BAND
4	DELAY/THRU						0	700	50	0	999	COAX	THRU
5													
6													
7	SHORT						17.544	700	50	0	999	COAX	SHORT(F)
8	OPEN	62	17	28	0		17.544	700	50	0	999	COAX	OPEN(F)

Step 2: Activating the Define Standard Menu.

1. Press Cal.
2. Press CAL KIT [...].
3. Press MODIFY [...].
4. Press DEFINE STANDARD.
5. Select standard number.

6. Select standard type.

If you did not select standard type as OPEN in step 2, skip to step 4.

Step 3: Entering a C Parameters.

1. Press `C0`. Then enter C0 ($\times 10^{-15}$ F).
2. Press `C1`. Then enter C1 ($\times 10^{-27}$ F/Hz).
3. Press `C2`. Then enter C2 ($\times 10^{-36}$ F/Hz^2).

Step 4: Entering an OFFSET Parameters.

1. Press `SPECIFY OFFSET`.
   - Press `OFFSET DELAY`. Then enter DELAY.
   - Press `OFFSET LOSS`. Then enter LOSS.
   - Press `OFFSET ZO`. Then enter Z_0.
2. Press `STD OFFSET DONE`.

Step 5: Entering a Standard Class Label.

1. Press `LABEL STD`.
2. Enter a standard label up to 10 characters.
3. Press `DONE`.

Step 6: Completing the Definition of a Calibration Kit.

1. Press `STD DONE (DEFINED)`.
2. Press `KIT DONE (MODIFIED)`.

To Define a Class Assignment

Step 1: Preparing for the Class Assignment.

1. Prepare the standard class assignment table for your calibration kit.

Table 4-3. Standard Class Assignment of the 85032B

	A	B	C	D	E	F	G	STANDARD CLASS LABEL
$S_{11}A$	2	8						OPENS
$S_{11}B$	1	7						SHORTS
$S_{11}C$	3							LOAD
$S_{22}A$	2	8						OPENS
$S_{22}B$	1	7						SHORTS
$S_{22}C$	3							LOAD
Forward Transmission	4							Fwd. Trans Thru
Reverse Transmission	4							Rev. Trans Thru
Forward Match	4							Fwd. Match Thru
Reverse Match	4							Rev. Match Thru
Response	1	7	2	8	4			RESPONSE
Response & Isolation	1	7	2	8	4			Response & Isol'n

Step 2: Specifying the Class.

1. Press `SPECIFY CLASS`.
2. Select standard class.

To Select	Press
$S_{11}A$	`SPECIFY: S11A`
$S_{11}B$	`S11B`
$S_{11}C$	`S11C`
$S_{22}A$	`SPECIFY: S22A`
$S_{22}B$	`S22B`
$S_{22}C$	`S22C`
Forward Transmission	`MORE` `SPECIFY: FWD. TRANS.`
Reverse Transmission	`MORE` `REV.TRANS.`
Forward Match	`MORE` `FWD. MATCH`
Reverse Match	`MORE` `REV.MATCH`
Response	`MORE` `RESPONSE`
Response & Isolation	`MORE` `RESPONSE & ISOL'N`

3. Enter the standard number from A to G.

4. Press CLASS DONE (SPEC'D).

Step 3: Labeling the Standard Class Label.

1. Press LABEL CLASS to label the standard class.
2. Select the standard class.
   See 2 of Step 2.
3. Enter or modify the correspondent standard class label.
4. Press LABEL DONE.

To Label and Save Calibration Kit.

1. Press LABEL KIT.
2. Enter label.
3. Press DONE KIT DONE (MODIFIED).
4. Press CAL KIT [...] SAVE USER KIT USER KIT.
5. Press RETURN.

After the user-defined calibration kit is defined, you can verify the definition using the copy function that lists standard parameters and class assignment.

To Verify Definition of User-Defined Calibration Kit.

1. Press (Cal) CAL KIT [...] USER KIT to specify the calibration kit as a user-defined kit.
2. Press (Copy) MORE CAL KIT DEFINITION.
   - To display the standard parameters defined, press STANDARD DEFINITON. Then press the softkey labeled standard number if you want to list the defined parameters.
   - To display the defined class assignment, press CLASS ASSIGNMENT.
3. To make a hardcopy, press PRINT [STANDARD].
4. To return to the normal display, press RESTORE DISPLAY.

Step 8: Reading the Measured Result

If you can obtain the correct trace on the display, you can readout the trace gain and frequency by using the marker. The analyzer can search for specific points (like peaks or ripples) by using the marker search function. This step provides procedures for reading values using the marker and the marker search functions.

- To read a value using the marker
- To stabilize the trace
- To use the sub-markers
- To use the Δmarker
- To search for a point which has the target value
- To search for the peak-to-peak of ripples using the statistics function
- To define the peak for search

To Read a Value Using the Marker

1. Press Marker.
2. Move the marker to the point you want to read using the  .
3. Read the marker value displayed on the upper right of the display.

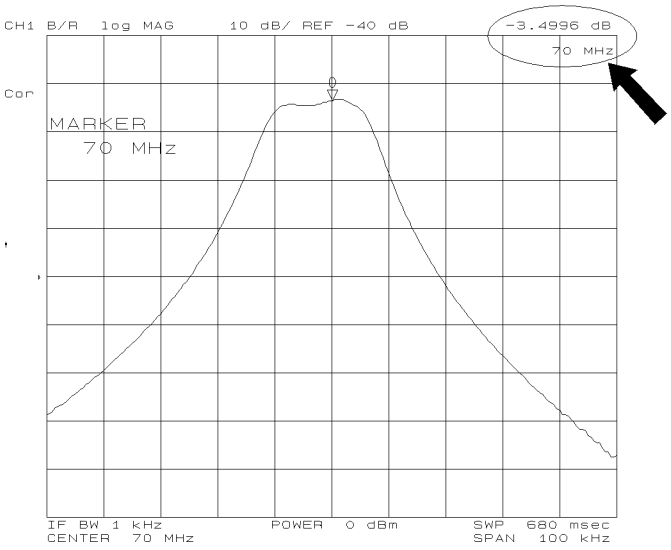


Figure 4-8. Marker Readout

To Stabilize the Trace

When the marker value changes frequently and it is difficult to read the value, you can use the following techniques to stabilize the trace:

- Stop the sweep.
- Use the averaging function.

To Stop the Sweep

1. Press (Trigger).
2. Press SWEEP: HOLD.

The sweep is stopped immediately (even if the sweep is in progress). If you want to restart the sweep, press CONTINUOUS.

To Reduce Variation of the Measurement Trace Using Averaging

1. Press (Bw/Avg).
2. Press AVERAGING FACTOR.
3. Enter the number of times to average. Then press (x1).
4. Toggle AVERAGING on OFF to ON off.

"Avg" appears on the left side of display when averaging is turned on. The averaging notation counts the averaging factor during averaging until it reaches the specified averaging factor. When averaging is completed, the counter halts and the trace continues updating with each sweep.

The averaging factor can be set from 0 to 999.

If you want to change the setting of any parameter when averaging, you can restart averaging from the 0 count.

To Restart the Averaging.

1. Press (Bw/Avg).
2. Press AVERAGING RESTART.

This resets the averaging counter to 0 and the counter starts incrementing with every sweep.

To Use the Sub-markers

1. Press (Marker).
2. Move the marker to a point where you want to set a sub-marker.
3. Press SUB MKR.
4. Select from SUB MKR 1 to 7.
5. Press (Utility).
6. Toggle MKR LIST on OFF to ON off.

The sub-marker appears at the point of that the marker is displayed. Sub-markers are fixed horizontally and you cannot move them.

The sub-marker value can only be displayed by using the marker list.

To clear a sub-marker, press (Marker) CLEAR SUB MKR. Then press the sub-marker number that you want to erase from the display.

To clear all the markers, press (Marker) PRESET MKRS.

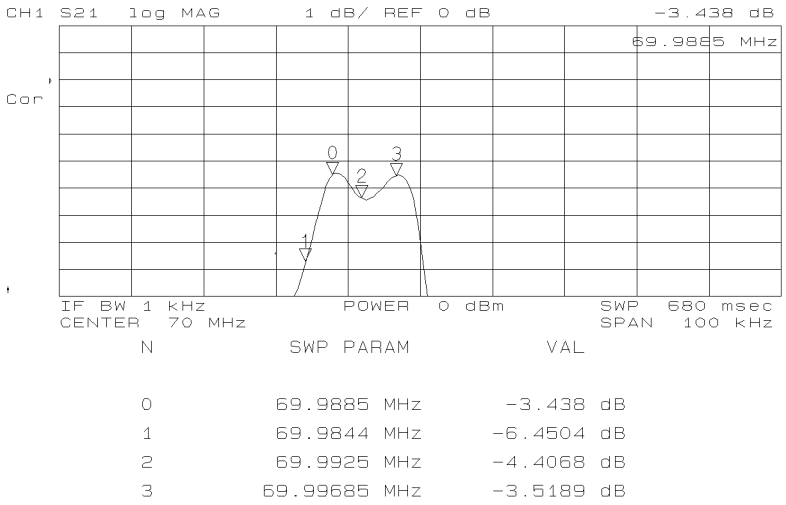


Figure 4-9. Sub-markers

To Use the ΔMarker

1. Press (Marker).
2. Place the marker on the point you want use as a reference point using the  .
3. Press ΔMODE MENU.
4. Press ΔMKR. The reference marker appears at the marker place.
5. Move the marker by using the following procedures:
   - Enter a difference frequency (or power) by using the numerical keys.
   - Turn the rotary knob until the marker moves to the point you want to read the value.
6. Read the gain and the frequency (or power) differences from the reference marker that are displayed on the upper right of the grid.

Figure 4-10. ΔMarker

To Search For a Point that has the Target Value

1. Press Search.
2. Press TARGET.
3. Enter the target value using 0 ... 9 and the unit terminator keys.

To search for a target on	Press
All of the display	TARGET
Left side of the marker	SEARCH LEFT
Right side of the marker	SEARCH RIGHT

When the Δmarker is active, the target value becomes the difference from the reference marker, not an absolute value. For example, you can search for the -3 dB cutoff point of a filter by mixing the Δmarker and the target search function.

To Search for the Peak-to-Peak of Ripples Using the Statistics Function

Step 1: To Specify the Search Range

1. Press (Marker). Then move the marker to the start point of the range.
2. Press `ΔMODE MENU` `ΔMKR` to place the reference marker on the start point of the range.
3. Move the marker to the end point of the range.
4. Press (Search) `SEARCH RANGE MENU`.
5. Toggle `PART SRCH on OFF` to `ON off`.
6. Press `MKRΔ→SEARCH RNG` to set the range defined by the reference marker and the marker as the search range.

Triangle-shaped indicator (△) at the bottom of the grid shows current search range.

Step 2: To Search For the Ripple

1. Press (Utility).
2. Toggle `STATISTICS on OFF` to `ON off`.

`STATISTICS` displays the mean value (`mean`), the standard deviation (`s.dev`), and the peak-to-peak value (`p-p`) of the ripple within the specified range of the active channel. This information is displayed on the upper right of the display (see Figure 4-11). If you did not specify the search range, the analyzer searches within the displayed area.

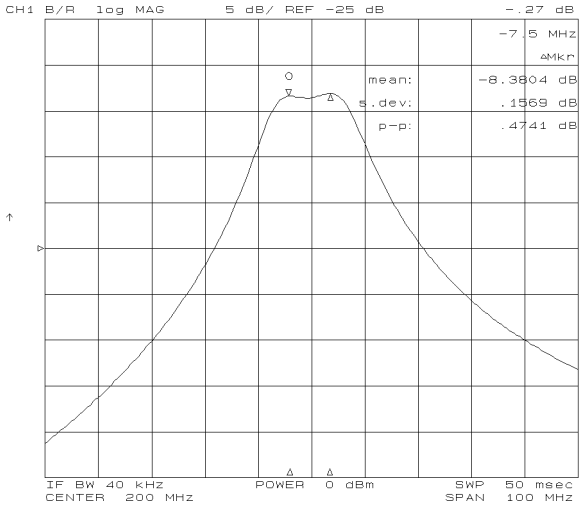


Figure 4-11. Ripple Parameters Readout

To Define the Peak for Search

You can define the target peak for the search function using the following techniques:

- Defining the peak slope to ignore the relatively broad peaks.
- Specifying the peak threshold to ignore the absolutely small peaks.

Defining the Peak Slope to Ignore the Relatively Broad Peaks

Entering Directly.

1. Press (Search) SEARCH:PEAK.
2. Press PEAK DEF MENU.
3. Press PEAK DEF:ΔX. Then enter a width of the peak.
4. Peak PEAK DEF:ΔY. Then enter a height of the peak.

Using the Marker.

1. Press (Marker). Then move the marker on the local maximum you want to search.
2. Press ΔMODE MENU ΔMKR
3. Press (Search) SEARCH:PEAK PEAK DEF MENU.
4. Move the marker to the foot of the peak.
5. Press MKR→PEAK DELTA.

This parameter defines the slope of the peak. The definition is made by defining ΔX and ΔY as shown in Figure 4-12. The search function searches only for peaks that are steeper than the specified slope. Use this function when the search function searches for a peak that has a gentle slope.

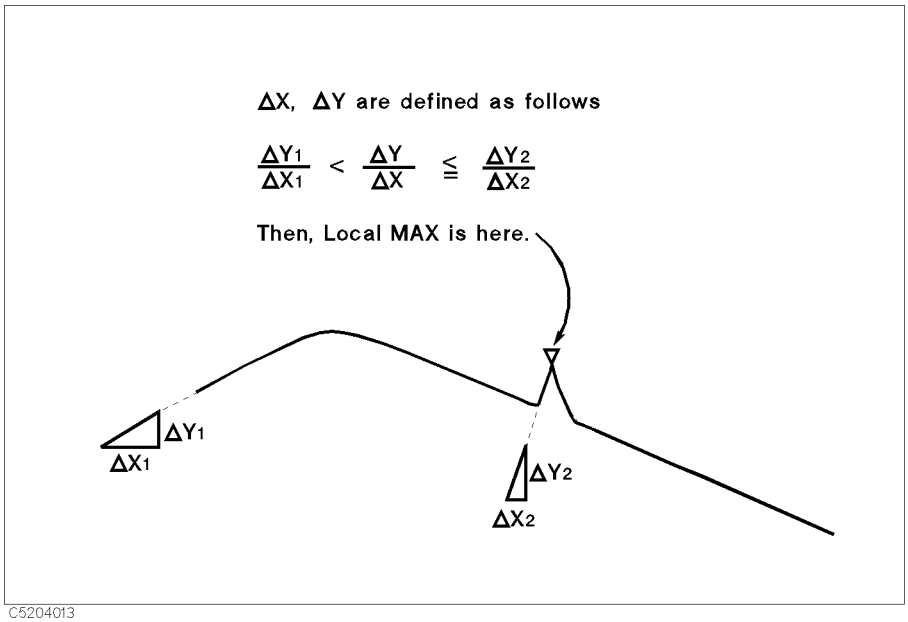


Figure 4-12. Peak Definition

Specifying the Peak Threshold to Ignore the Absolutely Small Peaks

Entering Directly.

1. Press (Search) `SEARCH:PEAK` `PEAK DEF MENU`.
2. Press `THRESHOLD VALUE`.
3. Enter a threshold value.
4. Toggle `THRESHOLD on OFF` to `ON off`.

The red threshold line is displayed. The all search function searches for only the upper side of the threshold line.

Using the Marker.

1. Press (Search).
2. Press `SEARCH:PEAK` `PEAK DEF MENU`.
3. Move the marker to the point you want to set as the threshold value.
4. Press `MKR→THRESHOLD`.
5. Toggle `THRESHOLD on OFF` to `ON off`.

For more information about a peak definition, see chapter 12 of the *Function Reference*.

5

Typical Network Measurements

This chapter provides the following typical measurement techniques using the network analyzer mode of operation:

- To measure the 3 dB bandwidth using the width function
- To monitor the spectrum of the network measurement input
- To measure the electrical length
- To measure the phase deviation
- To cancel an extension of a measurement cable

To Measure the 3 dB Bandwidth Using the Width Function

1. Press Search.
2. Do either one of the following:

To Use	Do
Maximum point as a reference	Press MAX.
Nominal frequency as a reference	Enter the nominal frequency using the numerical keys.

3. Press ΔMODE MENU ΔMKR to make the marker a reference.
4. Press Search WIDTH [OFF] WIDTH VALUE.
5. Press (—) 3 (×1) to enter −3 dB.
6. Toggle WIDTH on OFF to ON off.

The sub-markers appear at a specified level down on both sides of the reference marker and the center of the two sub-markers, as shown in Figure 5-1. The bandwidth (BW), center frequency (cent), Q factor (Q), insertion loss (loss), and left and right hand bandwidth from the center frequency (ΔL.F and ΔR.F) are displayed on the upper right of the grid.

You can move the reference marker using the rotary knob. When you enable the width function, the reference marker automatically turns into a tracking Δmarker that allows you to move the reference marker.

For more information about the width function, see chapter 12 of the *Function Reference* manual.

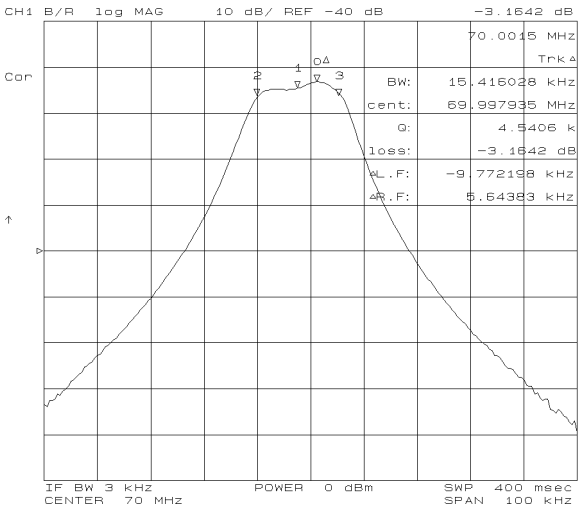


Figure 5-1.
Bandwidth Measurement Using Width Function

To Monitor the Spectrum of the Network Measurement Input

1. Press (Display). Then toggle DUAL CHAN on OFF to ON off.
2. Press (Ch 1) or (Ch 2) to activate the other channel (that is, the channel that was not used for the network measurement).
3. Press (Meas) ANALYZER TYPE.
4. Press SPECTRUM ANALYZER to set the spectrum analyzer mode.
5. Set the same frequency setting for both channels.
6. Press (Meas). Then select the input port (A or B) that you used for the network measurement.

You can monitor the spectrum of the network input without reconnecting the cables. This feature allows you to analyze the network characteristics while separating the effect of unknown signals (for example spurious signals) without performing any additional procedures.

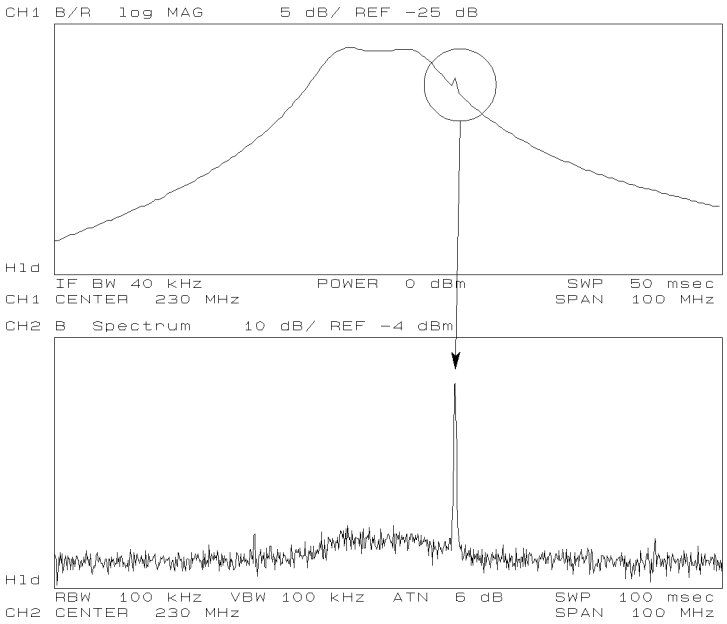
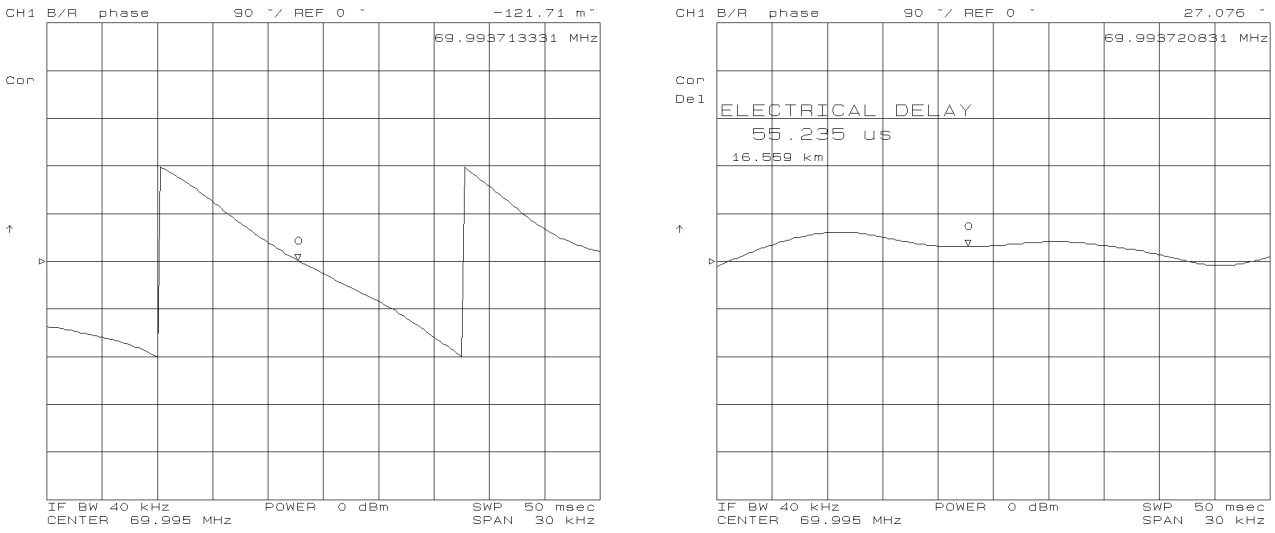


Figure 5-2. Spectrum Monitoring in the Network Measurement

To Measure the Electrical Length

1. Select the phase format.
2. Do one of the following procedure:
   - Using the marker:
     a. Press (Marker).
     b. Turn the rotary knob to position the marker at the center of the display.
     c. Press (Scale Ref) ELECTRICAL DELAY MENU.
     d. Press MKR→DELAY.
   - Using the rotary knob:
     a. Press (Scale Ref) ELECTRICAL DELAY MENU
     b. Press ELECTRICAL DELAY.
     c. Turn the rotary knob until the trace becomes flat at the interested frequency.
3. Press ELECTRICAL DELAY. Then read the electrical length that is displayed under the electrical delay time.



Before Adding the Electrical Length

After Adding the Electrical Length

Figure 5-3. Adding Electrical Length

If the average relative permitivity (ε_R) of the DUT is known over the frequency span, the length calculation can be adjusted to better indicate the actual length of the DUT. This can be done by entering the relative velocity factor for the DUT.

To Set Velocity Factor of a Cable

1. Press (Cal).
2. Press MORE.
3. Press VELOCITY FACTOR.
4. Enter a new value. Then press (×1).

The relative velocity factor for a given dielectric can be calculated by:

$$V_f = \frac{1}{\sqrt{\varepsilon_R}}$$

Default setting of the velocity factor is 1.

To Measure the Phase Deviation

To Display the Deviation from the Linear Phase

1. Set up the frequency range.
2. Display the phase trace by pressing (Format) PHASE.
3. Adjust the scale settings by pressing (Scale Ref) AUTO SCALE.
4. Press (Marker). Then move the marker to any of the points where the sloping trace crosses the center (place the marker on the sloping portion of the trace, not the vertical phase "wrap-around").
5. Press (Scale Ref) ELEC DELAY MENU MKR→DELAY to add enough electrical length to match the group delay present at the marker frequency.
6. Read the phase value as a deviation from the linear phase.

By adding the electrical length to flatten the phase response, the linear phase shift caused by the DUT is removed. The displayed response is the deviation from the linear phase.

To turn off the electrical length function, press (Scale Ref) ELEC DELAY MENU ELECTRICAL DELAY (0) (X1).

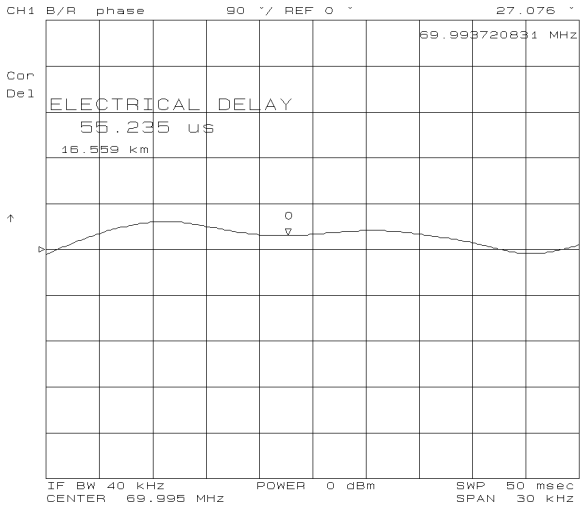


Figure 5-4. Deviation from the Linear Phase

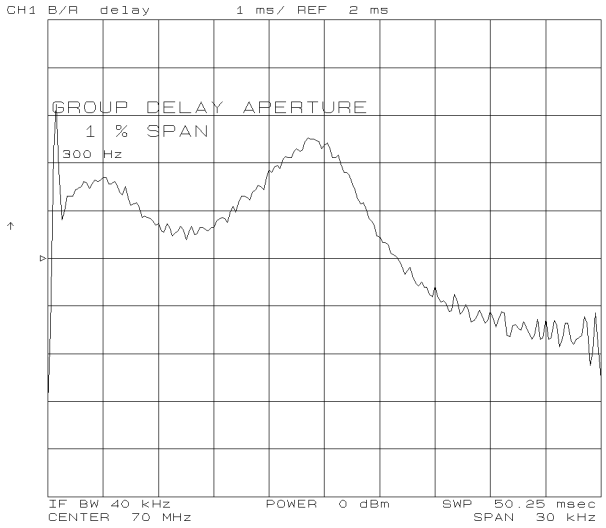
To Display the Group Delay

1. Press Format.
2. Press DELAY.

The group delay format displays phase deviation to group delay aperture. Therefore, setting the group delay aperture affects the trace shape. Setting a wider aperture makes the trace smoother. The default setting of the aperture is 1% of span.

1. Press Bw/Avg.
2. Press GROUP DELAY APERTURE.
3. Enter group delay aperture value as a percentage of the span.

The Group delay aperture is based on the number of points, not the real aperture. For example, if the number of points is 201, a 1% group delay aperture calculates the group delay using the adjacent measurement points on both sides. Therefore, the group delay trace is different by the number of points setting even if the group delay aperture is the same setting.



Aperture 1%

Aperture 5%

Figure 5-5. Setting Group Delay Aperture

For more information about a group delay, see chapter 12 of the *Function Reference*.

To Cancel an Extension of a Measurement Cable

If the Electrical Delay of the Extended Cable is Known

1. Press (Cal) `MORE` `PORT EXTENSION` to call the port extension menu.
2. Enter the electrical delay values for each input.
   - If you do not use the S-parameter test set:
     - Press `EXTENSION INPUT R`. Then enter the electrical delay of the cable that is connected to the R input.
     - Press `EXTENSION INPUT A`. Then enter the electrical delay of the A input.
     - Press `EXTENSION INPUT B`. Then enter the electrical delay of the B input.

     If you use the transmission/reflection (T/R) test set, enter the electrical delay of the cable that is connected to the TEST PORT (for the R and A inputs).

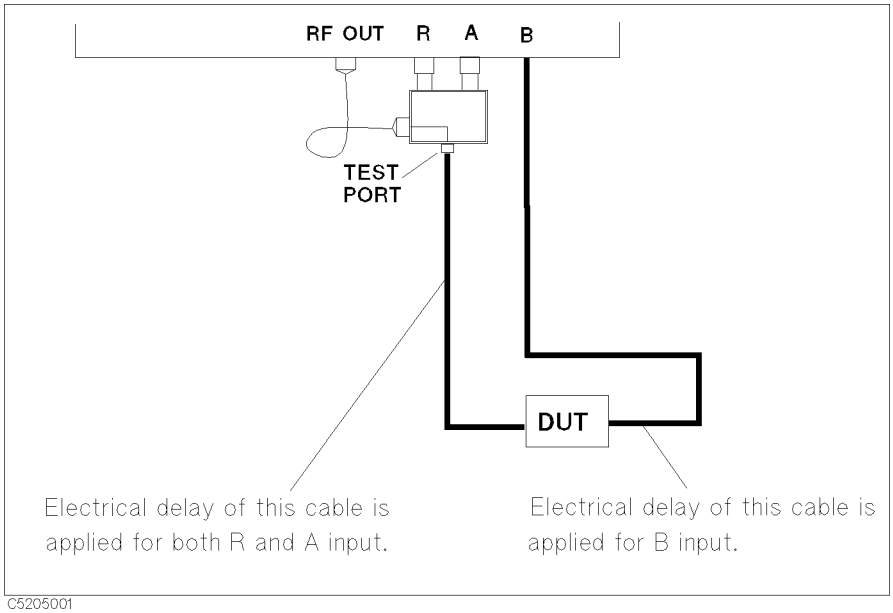


Figure 5-6. Port Extension With the T/R Test Set

   - If you use the S-parameter test set:
     - Enter "0" for `EXTENSION INPUT R`, `EXTENSION A` and `EXTENSION INPUT B` to clear the port extension of the R, A and B inputs.
     - Press `EXTENSION PORT 1`. Then enter the electrical delay of the PORT 1.

□ Press EXTENSION PORT 2. Then enter the electrical delay of the PORT 2.

3. Toggle EXTENSION on OFF to ON off to enable the port extension.

If the Electrical Delay of the Extended Cable is Unknown

You can determine the electrical delay of the cable by following methods:

- Measuring the electrical length of the cable.
- Measuring the OPENed or SHORTed cable reflection.

1. Connect the cable as shown in Figure 5-7.
2. Set up the frequency range to the measurement condition.
3. Press (Meas) B/R (or Trans:FWD S21 [B/R]) to select the transmission measurement.
4. Press (Format) PHASE to select the phase format.
5. Press (Marker). Then move the marker to the sloping trace that crosses the center of the display.
6. Press (Scale Ref) ELEC DELAY MENU ELECTRICAL DELAY MKR→DELAY, then read the electrical delay of the cable.
7. Press (0) (X1) to clear the electrical delay offset.
8. Enter a measured electrical delay as described in the "If the Electrical Delay of the Extended Cable is Known" procedure.

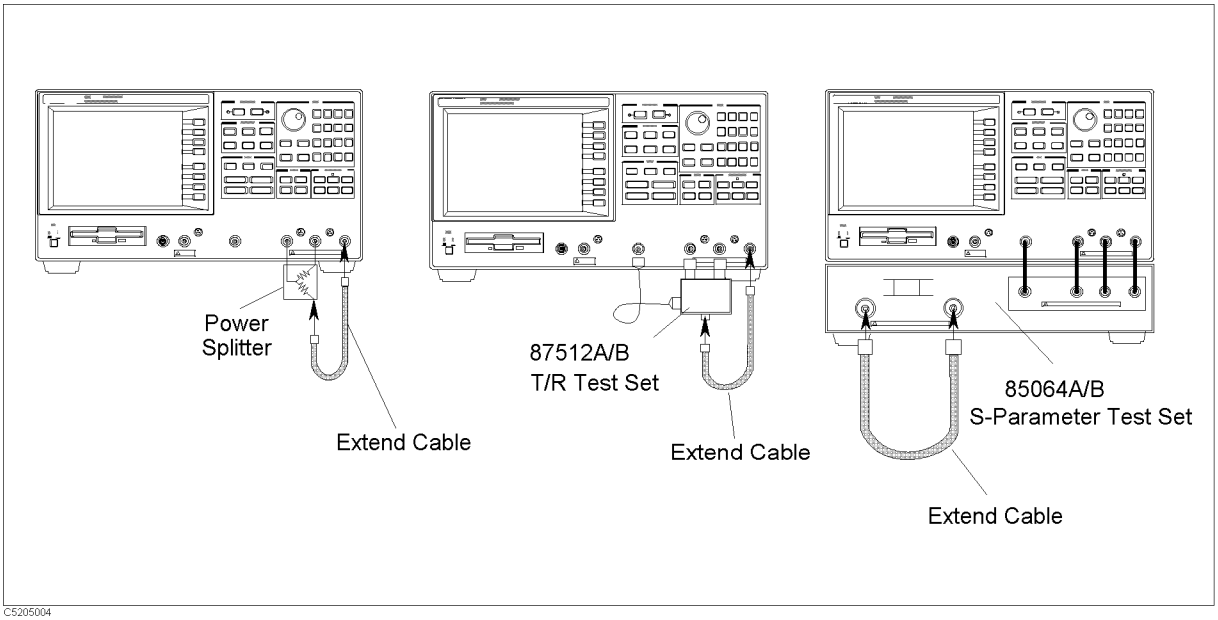


Figure 5-7. Cable Measurement Configuration

To Cancel an Extension of a Measurement Cable

1. Connect the cable as shown in Figure 5-8.
2. Set up the frequency range for the measurement condition.
3. Press (Meas) `A/R` (or `Refl:FWD S11 [A/R]`) to select the reflection measurement.
4. Press (Format) `PHASE` to select the phase format.
5. Press (Marker). Then move the marker to the sloping trace that crosses the center of the display.
6. Press (Scale Ref) `ELEC DELAY MENU` `ELECTRICAL DELAY` `MKR→DELAY`, then read the electrical delay of the cable. Note that this value is twice the real delay because there is an output and a return paths.
7. Press (0) (×1) to clear the electrical delay offset.
8. Enter *half the value* of the measured electrical delay as described in the "If the Electrical Delay of the Extended Cable is Known" procedure.

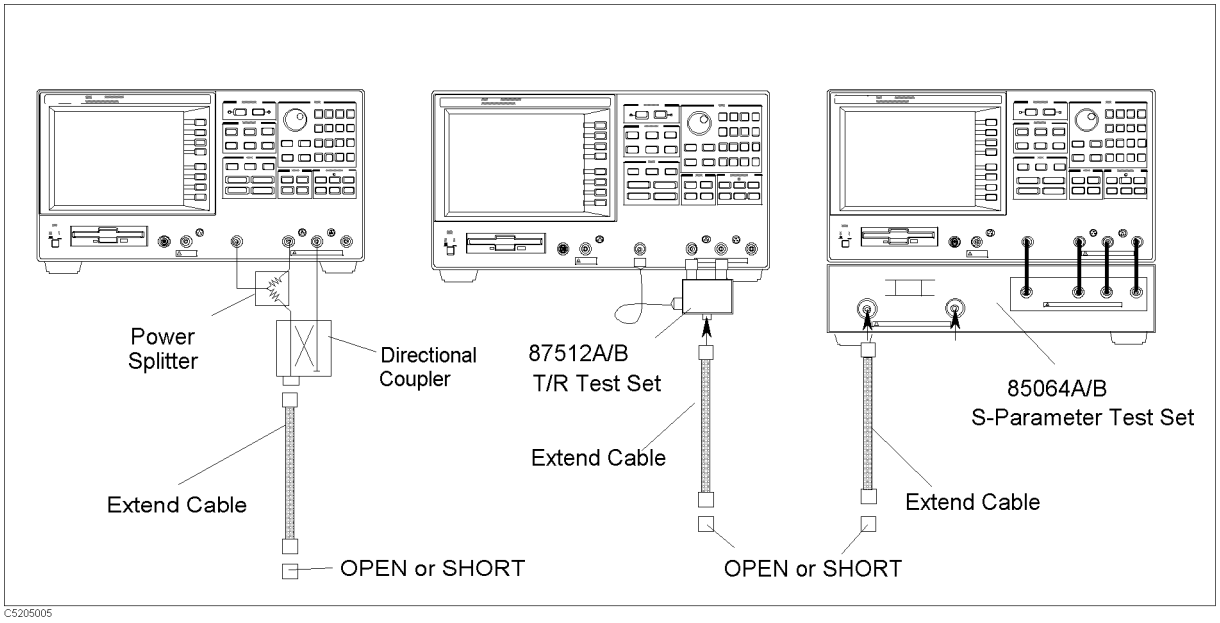


Figure 5-8. Cable Measurement Configuration

Notes For more information about Electrical delay and port extension, refer to chapter 5 of the *Function Reference*.

6

Using Features Common to Both Modes of Operation

This chapter provides procedures for the analyzer features that can be used in both the spectrum and network analyzer modes of operation.

This chapter provides the following procedures:

- To perform GO/NO-GO testing
- To offset the limit line
- To make a list sweep
- To use the trace memory
- To offset the trace
- To use the trace math functions
- To print and plot
- To save and recall

To Perform GO/NO-GO Testing

Step 1: Planning the Limit Lime

1. Determine the following parameters before editing the limit line:

Parameter	Description
Sweep Parameter	Frequency of each segment.
Upper Limit	Upper limit level of each segment.
Lower Limit	Lower limit level of each segment.

The limit line is constructed by connecting the segment points as shown in Figure 6-1.

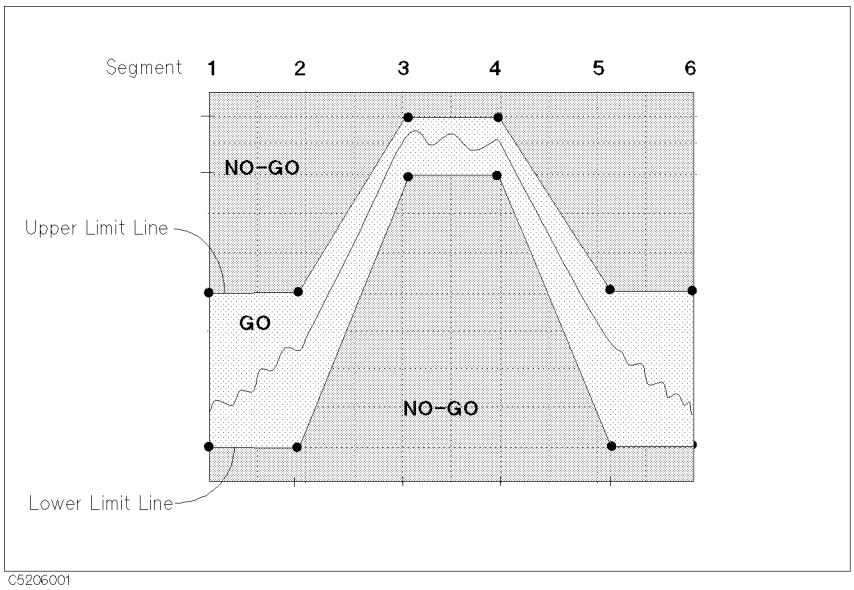


Figure 6-1. Limit Line Image

For example, if you want to specify four points for the limit test, the limit line image is as shown in Figure 6-2. Each point has frequency information and an upper and a lower limit value. Enter these values as described in the "Step 2: Editing a Limit Line Table" procedure.

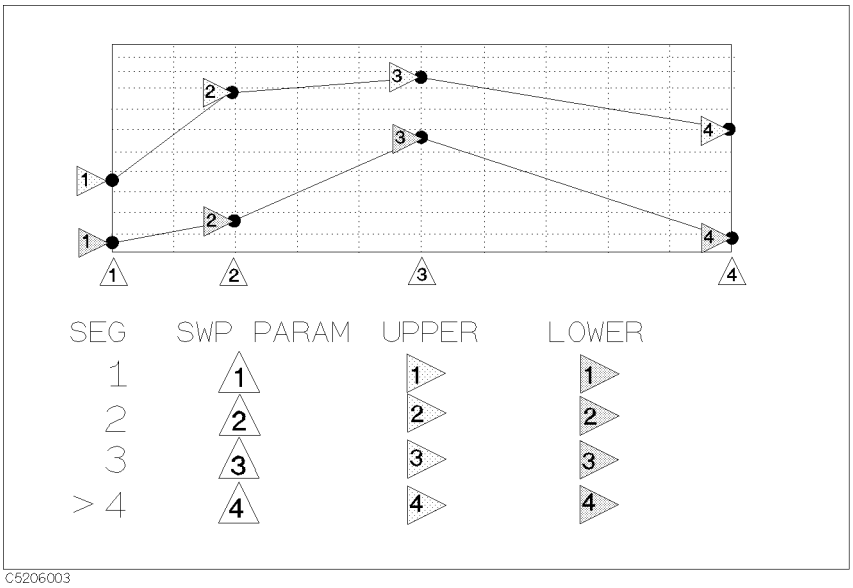


Figure 6-2. Frequency, Upper and Lower Limit

In this example, the limit line connects four limit points. If a measured trace exceeds the upper or lower limit line, the limit test fails.

Step 2: Editing a Limit Line Table

1. Set up the frequency range of the grid before starting the limit line edit.
2. Press System.
3. Press `LIMIT MENU`. Then toggle `LIMIT LINE on OFF` to `ON`. This makes it easier to understand the status of the limit line while you are editing it.
4. Press `EDIT LIMIT LINE` to call the limit line editor.
5. If an old limit line table is still in the limit line editor, press `CLEAR LIST` `CLEAR LIST YES` to clear it.
6. Press `EDIT` to edit the first segment.

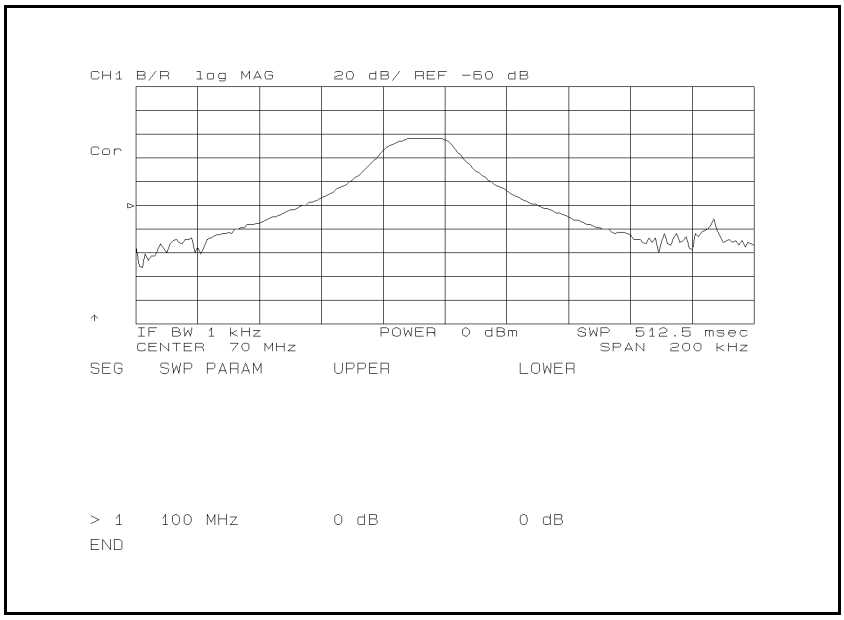


Figure 6-3. Limit Line Editor

7. Enter the frequency of the segment in one of the following ways:
   - Press `SWP PARAM`. Then enter the frequency of the segment.
   - Move the marker to the point you want to use as the frequency of the segment. Then press `MKR→SWP PARAM`.
8. Press `UPPER LIMIT`. Then enter a upper limit value.
9. Press `LOWER LIMIT`. Then enter a lower limit value.
10. Press `DONE` to end editing the segment.
11. Press `ADD` to edit the next segment.
12. Repeat list item 7 to list item 11 until all segments are defined.
13. When you finish editing all segments, press `DONE` to complete editing the limit line table.

You can enter the limit value using the middle and width method by pressing `MIDDLE VALUE` and `DELTA LIMIT`. You then enter the amplitude value as a middle amplitude value with a delta limit. The upper and lower limit lines appear at an equal positive and negative distance from the specified middle amplitude.

To Modify or Delete the Segment

1. Press `SEGMENT`.
2. Select the segment you want to delete or modify:
   - Enter the segment number you want to modify. Then press (X1).

- Move cursor, ">", to the segment you want to modify by using the ⇧, the ⇩, or the ◎.

3. Do either of the following:

To	Press
Modify specified segment	EDIT
Delete specified segment	DELETE

Step 3: Executing a Limit Line Test

To Make a Limit Line Test Active

1. Press System.
2. Press LIMIT MENU.
3. Toggle LIMIT TEST on OFF to ON off.

If the limit line test passes, a green PASS message appears on the right of the grid. If it fails, a red FAIL message is displayed. You can set the analyzer to beep if the limit line test fails. (See the "To Beep When the Limit Test is Failed" procedure.)

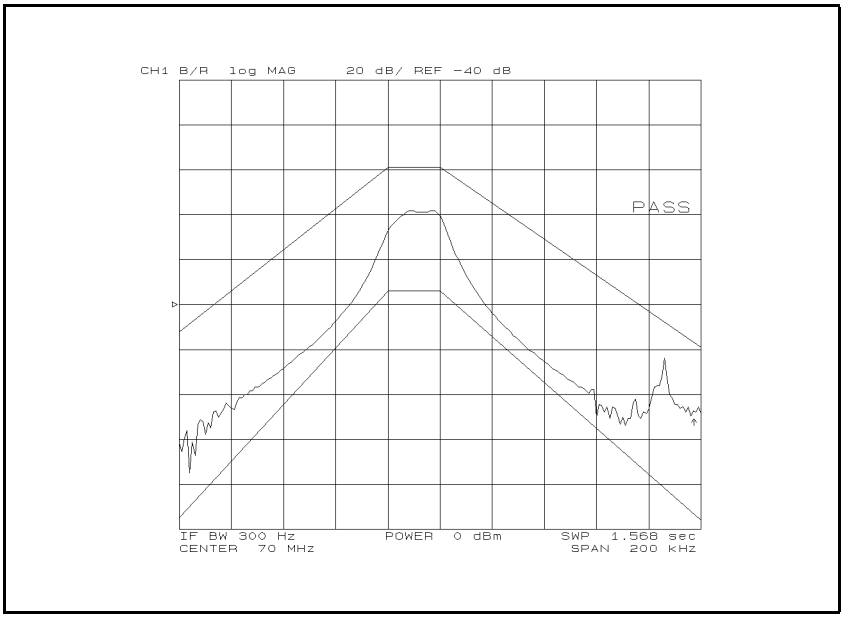


Figure 6-4. Limit Line Test

To Beep When the Limit Test is Failed

1. Press System.
2. Press LIMIT MENU.
3. Toggle BEEP FAIL on OFF to ON off.

Notes You can save the edited limit line table with all other instrument settings by pressing (Save) STATE. See "To Save an Analyzer Setting or Measurement Data" for more information.

To Offset the Limit Line

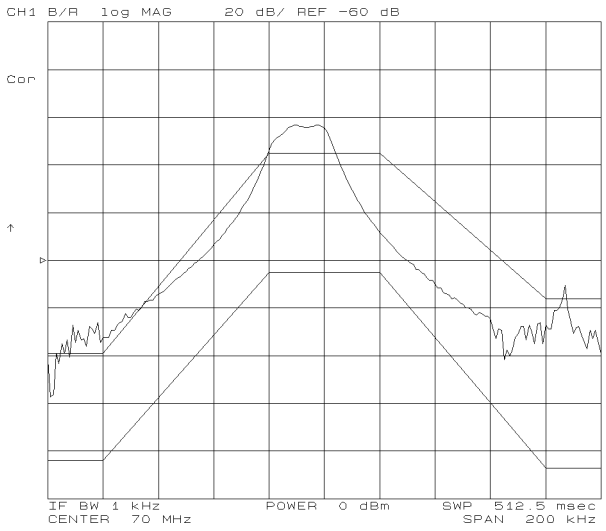
1. Press System LIMIT MENU.
2. Press LIMIT LINE OFFSETS.
3. Press the following keys:

To move line	Press
Horizontally	SWP PARAM OFFSET
Vertically	AMPLITUDE OFFSET

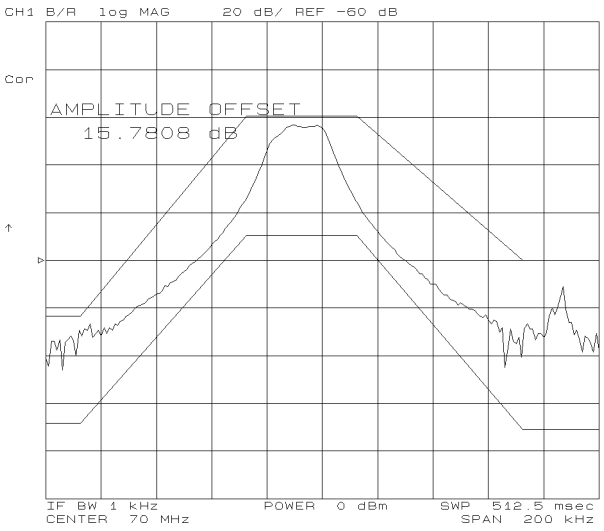
4. Then move the limit line by entering an offset value using one of the following:

To	Use
Move continuously	
Move with steps	⇧ ⇩
Enter offset value directly	0 ... 9 and unit keys

5. To move the limit line vertically to the marker position:
   a. Press Marker. Then move the marker to the point you want to set as the offset value.
   b. Press System LIMIT LINE MENU LIMIT LINE OFFSETS MKR→ AMP. OFS..
6. When you are finished offsetting the limit line, press RETURN.



Before Offset



After Offset

Figure 6-5. Offsetting Limit Lines

To clear the offset, enter 0 for all the offset values.

To Make a List Sweep

Step 1: Planning the List Sweep

1. Determine the following parameters before editing the sweep list.

Parameter	Description
Sweep Parameter	Frequency (or power for the power sweep) of each segment. Each segment cannot be continuous. You can reduce the sweep time by setting the segment only for the required part of the trace.
Number of points	Number of display points. You can adjust the display area for each segment by setting this parameter.
RBW	This parameter is for the spectrum analyzer mode. You can set the resolution bandwidth for the each segment individually. This is useful if you want to display higher resolution only for the specific segment.
IF BW	This parameter for the network analyzer mode. You can set the IF bandwidth for each segment individually. This is useful if you want to display higher dynamic range only for the specific segment.
Output Power	Output power from the RF OUT port of each segment. The allowable range is -70 dBm to +20 dBm. When you want to use the internal tracking generator for spectrum measurement, you can set this parameter in the spectrum analyzer mode. *

* RBW$\geq$10 kHz only.

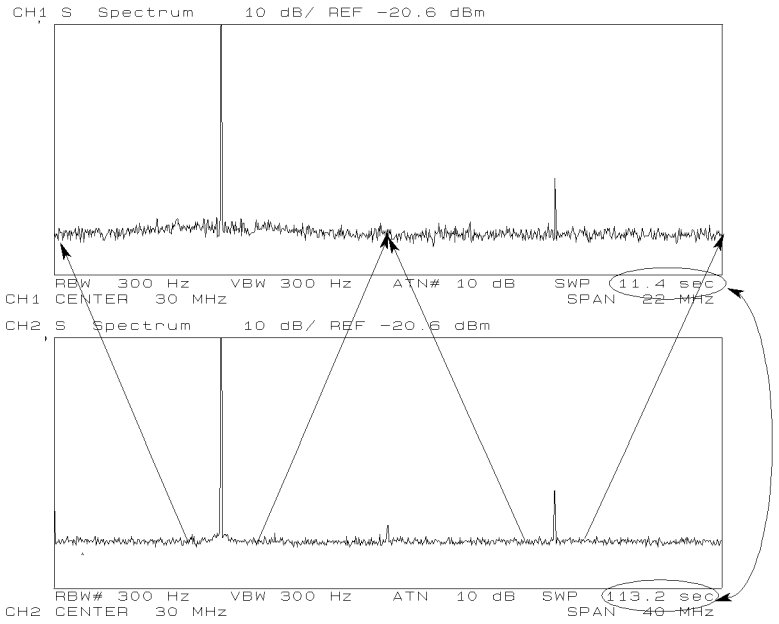


Figure 6-6. List Sweep (Upper) Reduces Sweep Time

Step 2: Editing a Sweep List

1. Press (Sweep) `SWEEP TYPE MENU` `EDIT LIST` to call the sweep list editor.
2. Press `EDIT` to edit the sweep list.

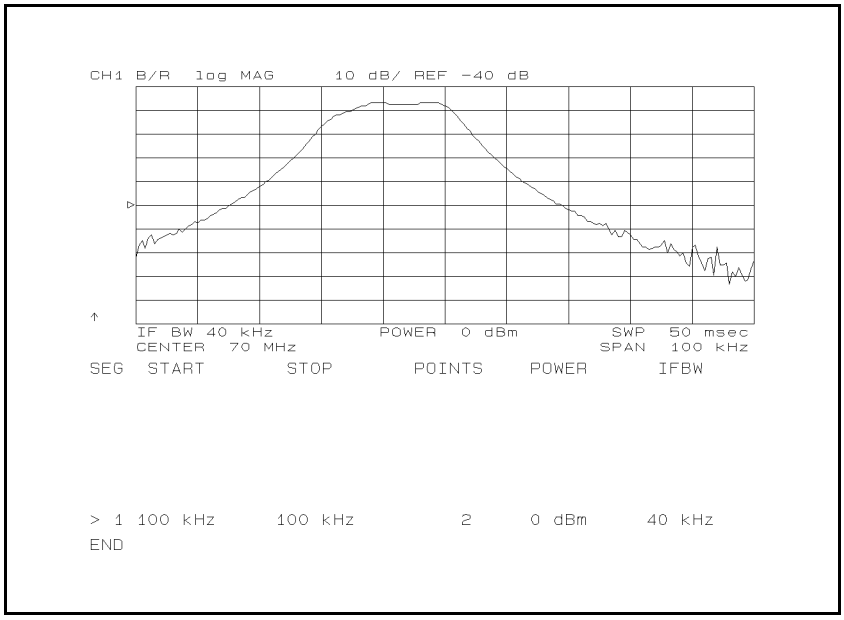


Figure 6-7. List Sweep Editor

3. Enter the frequency (or power) range of the segment.

- Move the marker to the start point. Then press SEGMENT: MKR→START.
- Move the marker to the stopping point. Then press MKR→STOP.
- Press (Start) to enter the start sweep parameter. Then press (Stop) to enter the stop sweep parameter.

4. Press NUMBER of POINTS. Then enter the number of points for the segment.
5. Set the output power:
   - When in spectrum analyzer mode, ignore this parameter unless you are using the internal tracking generator. The default value (0 dBm) is not applied until you turn on the RF Output.
   - Press POWER. Then enter the output power level for the segment.
6. Set the internal filter bandwidth:
   - When in spectrum analyzer mode, press RES BW to set the resolution bandwidth.
   - When in network analyzer mode, press IF BW to set the IF bandwidth.
7. Press SEGMENT DONE to complete editing the segment.
8. Press ADD to edit the next segment.
9. Repeat list item 3 to list item 8 until you complete editing of all required segments.
10. When you finish editing all the segments, press LIST DONE to complete the sweep list.

To Modify or Delete the Segment

1. Press SEGMENT.
2. Select the segment you want to delete or modify:
   - Enter the segment number you want to modify. Then press (×1).
   - Move the cursor, ">", to the segment you want to modify by using the (⇧), the (⇩), or the  .
3. Do either of the following:

To	Press
Modify specified segment	EDIT
Delete specified segment	DELETE

Step 3: Activating the List Sweep

1. Press (Sweep).
2. Press SWEEP TYPE MENU.
3. Press LIST FREQ.
4. If you use the output power in the spectrum analyzer mode, press (Source). Then toggle RF OUT on OFF to ON off.

Notes

If you want to delete an edited sweep list, press (Sweep) SWEEP TYPE MENU EDIT LIST CLEAR LIST CLEAR LIST YES.

You can save and recall the edited sweep list with all other instrument settings by pressing (Save) STATE. See "To Save an Analyzer Setting or Measurement Data" for more information.

For more information about the list sweep, see chapter 6 of the *Function Reference*.

To Use the Trace Memory

Step 1: To Store the Trace into the Trace Memory

1. Display the trace you want to store into the trace memory.
2. Press (Display).
3. Press DATA→MEMORY.

This operation only stores the digitized trace data into the trace memory (not the display on CRT). You can store the trace data for the trace memory of each channel individually. The stored trace data is retained until new data is stored, the analyzer is preset, or power is turned off.

Step 2: To Display Memory Traces

1. Press (Display).
2. Select the display trace:

To display	Press
Memory trace	MEMORY
Data and memory trace together	DATA and MEMORY
Data trace	DATA

Memory traces are displayed as green (channel 1) or red (channel 2) to distinguish between the two traces. You can change this color by using the modify colors menu under (Display) MORE ADJUST DISPLAY.

If the trace memory of the active channel is empty, the error message (CAUTION: NO VALID MEMORY TRACE) is displayed.

Notes

The scale of the memory trace is coupling with the data trace. If you want to change the scale setting for only the data or memory trace, toggle (Scale Ref) D&M SCALE [COUPLE] to [UNCOUPLE]. Then toggle SCALE FOR [DATA] or [MEMORY] before you change the settings.

For more information about trace memory, see chapter 5 of the *Function Reference*.

To Offset the Trace

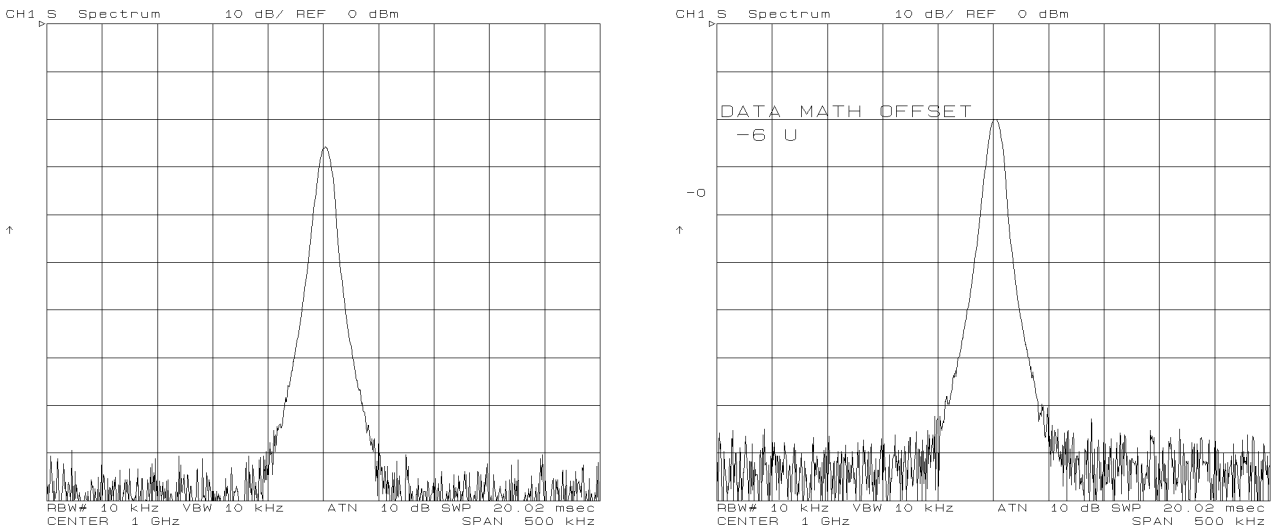
1. Press Display.
2. Press DATA MATH [DATA].
3. Press OFFSET. Then enter the offset value.

To	Press
Set the marker position to the offset value	MKR→OFFSET
Enter the offset value directly	OFFSET. Then enter the offset value

4. Press RETURN.

For example, if an attenuator is inserted between a signal source and the input port of the analyzer, the offset function corrects the attenuation of the input signal and displays the trace referenced to the input signal to the attenuator.

Offset indicator, -O, appears on the left side of the grid.



6 dB Attenuated Signal

After Offset

Figure 6-8. Offsetting the Trace

To Clear the Offset

1. Press Display.
2. Press DATA MATH [DATA-O].
3. Press DEFAULT GAIN & OFS.

Notes For more information about offsetting and multiplying traces, see the *Function Reference*.

To Use the Trace Math Function

1. Press (Display).
2. Press DATA MATH [DATA].
3. Do one of the following:

To	Press
Add Trace with Memory Trace	DATA+MEM
Subtract Trace with Memory Trace	DATA-MEM
Divide Trace with Memory Trace	DATA/MEM

To Turn Off the Data Math Function

1. Press (Display).
2. Press DATA MATH.
3. Press DATA MATH: DATA.

To Multiply the Trace

1. Press (Display).
2. Press DATA MATH [DATA].
3. Press GAIN. Then enter a multiplication factor:

To	Use
Change value continuously	
Change value with 1-2-5 steps	(↑) (↓)
Enter value directly	(0) . . . (9) and unit keys

To Clear a Multiplied Trace

1. Press (Display).
2. Press DATA MATH [G*DATA].
3. Press DEFAULT GAIN & OFS.

To Print

This step provides the following procedures for printing:

- To print out a display image
- To see or print a measured value list
- To print an analyzer setting

To Print Out a Display Image

1. Connect the printer to the analyzer with a cable.
2. Press (Copy) `PRINT [STANDARD]` to print out a display image.

To See or Print a Measured Value List

1. Press (Copy).
2. Press `MORE` `LIST VALUES` to display the measured value list.
   - To see all of the measured value list, press `NEXT PAGE` or `PREV PAGE` to turn the pages.
   - To print out the measured value list, press `PRINT [STANDARD]`.
   - To return to the measurement display, press `RESTORE DISPLAY`.

To Print an Analyzer Setting

1. Press (Copy).
2. Press `MORE` `OPERATING PARAMETERS` to display the analyzer setting table as shown below.
3. Press `PRINT [STANDARD]` to print out the settings.
4. To return to the measurement display, press `RESTORE DISPLAY`.

Analyzer Setting Table

OPERATING PARAMETER	CH1	CH2
ANALYZER TYPE	NA	NA
SWEEP TYPE	LIN FREQ	LIN FREQ
NUMBER of POINTS	201	201
RF OUT	ON	ON
PORT 1 ATTEN.	0 dB	0 dB
PORT 2 ATTEN.	0 dB	0 dB
GROUP DELAY APERTURE	1 % SPAN	1 % SPAN
PHASE OFFSET	0 °	0 °
PORT 1 EXTENSION	0 s	0 s
PORT 2 EXTENSION	0 s	0 s
INPUT R EXTENSION	0 s	0 s
INPUT A EXTENSION	0 s	0 s
INPUT B EXTENSION	0 s	0 s
	OFF	OFF
VELOCITY FACTOR	1	1
CAL KIT	7mm	7mm
Z0	50 ohm	50 ohm
CAL TYPE	NONE	NONE

To Save and Recall

This step provides the following procedures for saving and recalling:

- To save an analyzer setting or measurement data
- To recall a saved analyzer setting
- To save a display image to a TIF file
- To save measured data for a spreadsheet
- To copy the file
- To initialize a disk for use
- To initialize the RAM disk for use

To Save an Analyzer Setting or Measurement Data

1. Insert a LIF or DOS formatted 3.5 inch disk into the built-in disk drive (if you are recalling an instrument state file from the RAM disk, skip this step).
2. Press (Save).
3. Select a save data type:

Save Data Type	Press
Instrument states only	STATE
Measurement data only[1]	DATA ONLY (binary)

1 You can specify a data array type. See the "To Specify a Data Array Type" procedure.

4. Select where the file is stored by pressing either STOR DEV [DISK] (for the built-in disk drive) or STOR DEV [MEMORY] (for the RAM disk).
5. Enter a filename. Then press DONE.

The analyzer automatically detects the disk format as either the LIF (Logical Interchange Format) or DOS (Disk Operating System). If you insert an any other format type disk, an error message is displayed.

For more information, appendix C of the *Function Reference* provides a complete list of the instrument state to be saved.

Note

You can automatically recall the instrument state every time the analyzer is turned ON. Use "AUTOREC" as a filename. The analyzer recalls this file from a disk that is inserted in the built-in disk drive when it is turned ON.

To Specify a Data Array Type

1. Press (Save) DEFINE SAVE DATA.
2. Toggle to ON off the item what you want to save:

Data array	Toggle
Raw data array	RAW on OFF to ON off
Calibration data array	CAL on OFF to ON off
Data array	DATA on OFF to ON off
Memory array	MEM on OFF to ON off
Data Trace array	DATA TRACE on OFF to ON off
Memory Trace array	MEM TRACE on OFF to ON off

3. Press RETURN to return to the top menu.

To Recall a Saved Analyzer Setting

1. Insert a disk (if you are recalling an instrument state file from the RAM disk, skip this step).
2. Press (Recall).
3. Select where the file is stored by pressing either STOR DEV [DISK] (for a built-in disk drive) or STOR DEV [MEMORY] (for a RAM disk).
4. Search for the filename you want to recall (the files are listed on the softkey label).
5. If a target file is not listed on the softkey label, turn the label page by pressing PREV FILES or NEXT FILES.
6. Press the softkey corresponding to the filename label.

To Save a Display Image to a TIFF File

1. Press (Save) GRAPHICS.
2. Select where to store the file by pressing either STOR DEV [DISK] (for a built-in disk drive) or STOR DEV [MEMORY] (for a RAM disk).
3. Enter filename. Then press DONE.

Note

A display image is saved according to the color setup you have done on the print setup menu ((Copy) PRINT SETUP). You can choose from PRINT:STANDARD (black and white), PRINT COLOR [FIXED] (color against white background), and PRINT COLOR [VARIABLE] (color against black background).

The analyzer saves a TIFF file with an extension, ".TIF" for a DOS format, or a suffix, "_T" for a LIF format.

If there is a file that has the same name you entered on the disk, the error message, "filename error" will be displayed. To save the file, use the other filename to save or purge the old file. To purge a file,

press (Save) FILE UTILITIES PURGE FILE then select the displayed filename by pressing the associated softkey.

To Save Measured Data for a Spreadsheet

1. Insert a DOS format disk into the built-in disk drive.
2. Press (Save) DATA ONLY.
3. Press SAVE ASCII.
4. Select the built-in disk drive as the storage device by toggling to STOR DEV [DISK].
5. Enter a filename. Then press DONE.

The analyzer saves an ASCII file with a ".TXT" extension.

The measured data is saved as ASCII text. Each value is separated by a tab. When you open this file from the spreadsheet software, specify the file format as the "TEXT with TAB delimiter".

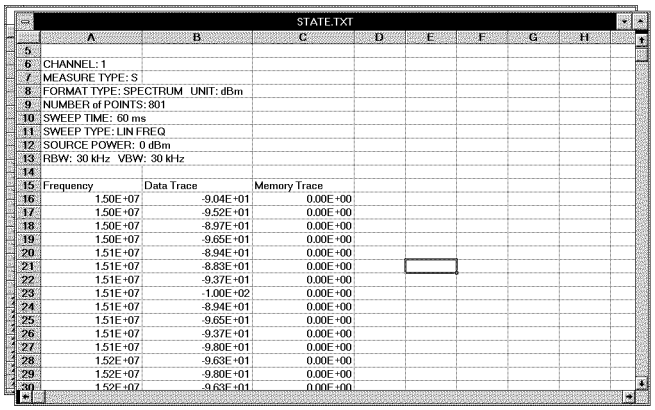


Figure 6-9. Reading Saved Data from Spreadsheet Software

To Copy the File

1. Press (Save) FILE UTILITIES.
2. Press COPY FILE.
3. Select a storage device where the file is stored by toggling either STOR DEV [DISK] (for the build-in disk drive) or STOR DEV [MEMORY] (for the RAM disk).
4. Search for the filename you want to recall (the files are listed on the softkey label).
5. If a target file is not listed on the softkey labels, turn the label page by pressing PREV PAGE or NEXT PAGE.
6. Press the softkey corresponding to the filename label.
7. Enter the filename of the target file.

8. Select the target storage device by toggling `STOR DEV [DISK]` or `[MEMORY]`.
9. Press `DONE` to copy the file.

You cannot copy a file between the LIF and DOS formats. When you want to copy a file on a DOS formatted disk to the RAM disk, you must initialize the RAM disk to the DOS format.

To Initialize a Disk for Use

Note 

Initializing the disk erases all data on the disk.

1. Verify that the disk is not write protected.
2. Insert the disk.
3. Press (Save) `FILE UTILITIES`.
4. Select the disk format (either DOS or LIF) by toggling `FORMAT [DOS]` or `[LIF]`.
5. Toggle to `STOR DEV [DISK]` to select the disk drive.
6. Press `INITIALIZE`.
7. Press `INITIALIZE DISK: YES` to initialize the disk.

To Initialize the RAM Disk for Use

Note 

Initializing the RAM disk erases all data on the RAM disk.

1. Press (Save) `FILE UTILITIES`.
2. Select the disk format (either DOS or LIF) by toggling `FORMAT [DOS]` or `[LIF]`.
3. Toggle to `STOR DEV [MEMORY]` to select the RAM disk.
4. Press `INITIALIZE INITIALIZE MEMORY: YES`.

The files on the RAM disk are kept until the analyzer is turned off. When the analyzer is turned off, the data on the RAM disk is lost.

The format is retained in the backup memory. The analyzer initializes the RAM disk automatically every time it is turned ON (using the format in the backup memory).

Notes For more information about saving data, see chapter 8 of the *Function Reference*.

A

Manual Changes

Introduction

This appendix contains the information required to adapt this manual to earlier versions or configurations of the 4396B than the current printing date of this manual. The information in this manual applies directly to the 4396B serial number prefix listed on the title page of this manual.

Manual Changes

To adapt this manual to your 4396B, see Table A-1 and Table A-2, and make all the manual changes listed opposite your instrument's serial number and firmware version.

Instruments manufactured after the printing of this manual may be different from those documented in this manual. Later instrument versions will be documented in a manual changes supplement that will accompany the manual shipped with that instrument. If your instrument's serial number or ROM version is not listed on the title page of this manual, in Table A-1, or Table A-2, make changes according to the *yellow MANUAL CHANGES* supplement.

In additions to information on changes, the supplement may contain information for correcting errors (Errata) in the manual. To keep this manual as current and accurate as possible, Agilent Technologies recommends that you periodically request the latest *MANUAL CHANGES* supplement.

For information concerning serial number prefixes not listed on the title page or in the *MANUAL CHANGE* supplement, contact the nearest Agilent Technologies office.

To confirm the firmware version, turn ON the power for the 4396B or execute `*IDN?` on the external controller.

Table A-1. Manual Changes by Serial Number

Serial Prefix or Number	Make Manual Changes

Table A-2. Manual Changes by ROM Version

Version	Make Manual Changes

Serial Number

Agilent Technologies uses a two-part, ten-character serial number that is stamped on the serial number plate (see Figure A-1) attached to the rear panel. The first five characters are the serial prefix and the last five digits are the suffix.

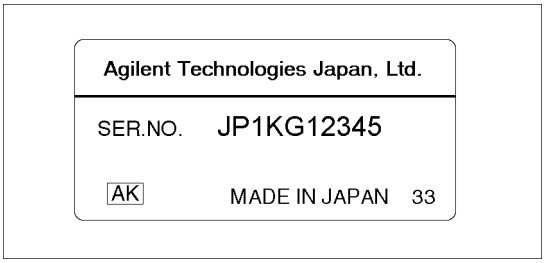


Figure A-1. Serial Number Plate

Index

REGIONAL SALES AND SUPPORT OFFICES

For more information about Agilent Technologies test and measurement products, applications, services, and for a current sales office listing, visit our web site: http://www.agilent.com/find/tmdir. You can also contact one of the following centers and ask for a test and measurement sales representative. *11/29/99*

United States:
Agilent Technologies
Test and Measurement Call Center
P.O.Box 4026
Englewood, CO 80155-4026
(tel) 1 800 452 4844

Canada:
Agilent Technologies Canada Inc.
5150 Spectrum Way
Mississauga, Ontario
L4W 5G1
(tel) 1 877 894 4414

Europe:
Agilent Technologies
Test & Measurement
European Marketing Organization
P.O.Box 999
1180 AZ Amstelveen
The Netherlands
(tel) (31 20) 547 9999

Japan:
Agilent Technologies Japan Ltd.
Call Center
9-1, Takakura-Cho, Hachioji-Shi,
Tokyo 192-8510, Japan
(tel) (81) 426 56 7832
(fax) (81) 426 56 7840

Latin America:
Agilent Technologies
Latin American Region Headquarters
5200 Blue Lagoon Drive, Suite #950
Miami, Florida 33126
U.S.A.
(tel) (305) 267 4245
(fax) (305) 267 4286

Australia/New Zealand:
Agilent Technologies Australia Pty Ltd
347 Burwood Highway
Forest Hill, Victoria 3131
(tel) 1-800 629 485 (Australia)
(fax) (61 3) 9272 0749
(tel) 0 800 738 378 (New Zealand)
(fax) (64 4) 802 6881

Asia Pacific:
Agilent Technologies
24/F, Cityplaza One, 1111 King's Road,
Taikoo Shing, Hong Kong
(tel) (852)-3197-7777
(fax) (852)-2506-9284