Measurement Guide and Programming Examples
N9060A Spectrum Analyzer Measurement Application

For use with N9020A MXA Signal Analyzer

This manual also provides documentation for the N9060A IQ Analyzer Measurement Application

Agilent Technologies

Manufacturing Part Number: N9060-90001

Printed in USA

October 2006

Notice

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

Agilent Technologies makes no warranty of any kind with regard to this material, including but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Safety Information

The following safety symbols are used throughout this manual. Familiarize yourself with the symbols and their meaning before operating this instrument.

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

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

NOTE *Note* calls out special information for the user's attention. It provides operational information or additional instructions of which the user should be aware.

Where to Find the Latest Information

Documentation is updated periodically. For the latest information about Agilent Technologies MXA Signal Analyzer, including firmware upgrades and application information, please visit the following Internet URL:

http://www.agilent.com/find/mxa

Contents

Contents

1 Getting Started with the Measurement Application

This chapter provides some basic information about using the Spectrum Analyzer and IQ Analyzer Measurement Application Modes. It includes topics on:

- "Making a Basic Measurement" on page 8
- "Recommended Test Equipment" on page 13
- "Accessories Available" on page 14

Technical Documentation Summary:

The N9020A MXA Signal Analysis measurement platform:	
Getting Started	Turn on process, Windows XP use/configuration, Front and rear panel
Specifications	Specifications for all available Measurement Applications and optional hardware (for example, Spectrum Analyzer and W-CDMA)
Functional Testing	Quick checks to verify overall instrument operation.
Instrument Messages	Descriptions of displayed messages of Information, Warnings and Errors
Measurement Application specific documentation: (for example, Spectrum Analyzer Measurement Application or W-CDMA Measurement Application)	
Measurement Guide and Programming Examples	Examples of measurements made using the front panel keys or over a remote interface. The programming examples use a few different programming languages, and copies of the executable files are available.
User's/Programmer's Reference	Descriptions of front panel key functionality and the corresponding SCPI commands. May also include some concept information.

Making a Basic Measurement

Refer to the description of the instrument front and rear panels to improve your understanding of the Agilent MXA Signal Analyzer measurement platform. This knowledge will help you with the following measurement example.

This section includes:

- "Using the Front Panel" on page 9
- "Presetting the Spectrum Analyzer" on page 10
- "Viewing a Signal" on page 10

CAUTION Make sure that the total power of all signals at the analyzer input does *not* exceed +30 dBm (1 watt).

Using the Front Panel

Entering Data

When setting measurement parameters, there are several ways to enter or modify the value of the *active* function:

Knob	Increments or decrements the current value.
Arrow Keys	Increments or decrements the current value.
Numeric Keypad	Enters a specific value. Then press the desired terminator (either a unit softkey, or the **Enter** key).
Unit Softkeys	Terminate a value that requires a unit-of-measurement.
Enter Key	Terminates an entry when either no unit of measure is needed, or you want to use the default unit.

Using Menu Keys

Menu Keys (which appear along the right side of the display) provide access to many analyzer functions. Here are examples of menu key types:

Toggle — Allows you to activate/deactivate states.

Exampl e: [Signal On] — Toggles the selection (underlined choice) each time you press the key.

Submenu — Displays a new menu of softkeys.

Exampl e: [Catalo] — A submenu key allows you to view a new menu of softkeys related to the submenu key category.

Choice — Allows you to make a selection from a list of values.

Exampl e: [Y Axis] — A choice key displays the currently selected submenu choice, in this example, dBm. When the choice is made, the submenu automatically returns.

Adjust — Highlights the softkey and sets the active function.

Exampl es: [CF 13.2550000] — Press this type of key and enter a value.

[Attenua 10.00 Auto] — The default for softkeys with an automatic (**Auto**) or manual (**Man**) choice is automatic. *After* you enter a value, the selection changes to manual. You can also press the softkey twice to change to manual.

Presetting the Spectrum Analyzer

The preset function provides a known starting point for making measurements. The analyzer has two main types of preset:

User Preset Restores the analyzer to a user-defined state.

Mode Preset This type of preset restores the currently selected mode to a default state.

For details, see the help or User's and Programmer's Reference.

Creating a User Preset

If you constantly use settings which are not the normal defaults, use the following steps to create a user-defined preset:

1. Set analyzer parameters as desired.
2. Press **User Preset**, **Save User Preset** to set the current parameters as the user preset state.
3. Then press **User Preset**, **User Preset** when you want to select the preset state.

Viewing a Signal

1. Press **Mode Preset** to return the current mode settings to its factory defaults.
2. Press **Input/Output**, **RF Calibrator**, **50 MHz**. to route the internal 50 MHz signal to the analyzer input.
3. Press **AMPTD Y Scale**, 10 dBm to set the reference level to 10 dBm.
4. Press **FREQ Channel**, **Center Freq**, 40 MHz to set the center frequency to 40 MHz.

   The 50 MHz reference signal appears on the display.
5. Press **SPAN**, 50 MHz to set the frequency span to 50 MHz.

TIP If the signal is on screen, press **FREQ**, **Auto Tune** to center and zoom in on the signal.

Reading Frequency & Amplitude

6. Press **Peak Search**.

   This activates a marker and places it on the highest amplitude signal.

   Note that the frequency and amplitude of the marker appear in the active function block in the upper-right of the display. You can use the knob, the arrow keys, or the softkeys in the Peak Search menu to move the marker around on the signal.

7. If you have moved the marker, return it to the peak of the 50 MHz signal by pressing **Peak Search** again.

Changing Reference Level

8. Press **AMPLTD Y Scale** and note that reference level (`Ref Level`) is now the active function. Press **Marker →**, **Mkr → Ref Lvl**.

   Note that changing the reference level changes the amplitude value of the top graticule line.

Improving Frequency Accuracy

9. To increase the accuracy of the frequency reading in the marker annotation, turn on the frequency count function.

   - Press **Marker**, **More**, **Marker Count**.

     The **Marker Count** softkeys appear.

   Note softkey **Counter On Off**. If **Off** is underlined, press the softkey to toggle marker count on.

   - The marker active function annotation changes from `Mkr1` to `Cntr1`.
   - The displayed resolution in the marker annotation improves.

NOTE When you use the frequency count function, if the ratio of the resolution bandwidth to the span is less than 0.002, you would get a display message that you need to reduce the Span/RBW ratio. This is because the resolution bandwidth is too narrow.

10. Press **Marker →**, **Mkr → CF** to move the 50 MHz peak to the center of the display.

Valid Marker Count Range

11. Move the marker down the skirt of the 50 MHz peak. Note that although the readout in the active function changes, as long as the marker is at least 26 dB above the noise, the counted value (upper-right corner of display) does not change. For an accurate count, the marker does not have to be exactly at the displayed peak.

NOTE Marker count functions properly only on CW signals or discrete peaks. For a valid reading, the marker must be ≥26 dB above the noise.

12. Press **BW**, **Res BW**, then enter a new value. This action makes the resolution bandwidth (RBW) the active function and allows you to experiment with different resolution bandwidth values.

13. Press **Marker**, **Off** to turn the marker off.

Recommended Test Equipment

The following table list the test equipment you will need to perform the example measurements describe in this manual.

NOTE To find descriptions of specific analyzer functions, for the N9060A Spectrum Analyzer Measurement Application, refer to the *Agilent Technologies User's and Programmer's Reference*.

Test Equipment	Specifications	Recommended Model
Signal Sources		
Signal Generator (2)	0.25 MHz to 4.0 GHz Ext Ref Input	E443XB series or E4438C
Adapters		
Type-N (m) to BNC (f) (6)		1250-0780
Cables		
BNC, 122-cm (48-in) (3)		10503A
Miscellaneous		
Directional Bridge		86205A

Accessories Available

A number of accessories are available from Agilent Technologies to help you configure your analyzer for your specific applications. They can be ordered through your local Agilent Sales and Service Office and are listed below.

NOTE There are also some instrument options available that can improve your measurements. Some options can only be ordered when you make your original equipment purchase. But some are also available as kits that you can order and install later. Order kits through your local Agilent Sales and Service Office.

For the latest information on Agilent MXA Signal Analyzer options and upgrade kits, visit the following Internet URL:http://www.agilent.com/find/sa_upgrades

50 Ohm Load

The Agilent 909 series of loads come in several models and options providing a variety of frequency ranges and VSWRs. Also, they are available in either 50 ohm or 75 Ohm. Some examples include the:

909A: DC to 18 GHz
909C: DC to 2 GHz
909D: DC to 26.5 GHz

50 Ohm/75 Ohm Minimum Loss Pad

The HP/Agilent 11852B is a low VSWR minimum loss pad that allows you to make measurements on 75 Ohm devices using an analyzer with a 50 Ohm input. It is effective over a frequency range of dc to 2 GHz.

75 Ohm Matching Transformer

The HP/Agilent 11694A allows you to make measurements in 75 Ohm systems using an analyzer with a 50 Ohm input. It is effective over a frequency range of 3 to 500 MHz.

AC Probe

The Agilent 85024A high frequency probe performs in-circuit measurements without adversely loading the circuit under test. The probe has an input capacitance of 0.7 pF shunted by 1 MOhm of resistance and operates over a frequency range of 300 kHz to 3 GHz. High probe sensitivity and low distortion levels allow measurements to be made while taking advantage of the full dynamic range of the signal analyzer.

AC Probe (Low Frequency)

The Agilent 41800A low frequency probe has a low input capacitance and a frequency range of 5 Hz to 500 MHz.

Broadband Preamplifiers and Power Amplifiers

Preamplifiers and power amplifiers can be used with your signal analyzer to enhance measurements of very low-level signals.

- The Agilent 8447D preamplifier provides a minimum of 25 dB gain from 100 kHz to 1.3 GHz.
- The Agilent 87405A preamplifier provides a minimum of 22 dB gain from 10 MHz to 3 GHz. (Power is supplied by the probe power output of the analyzer.)
- The Agilent 83006A preamplifier provides a minimum of 26 dB gain from 10 MHz to 26.5 GHz.
- The Agilent 85905A CATV 75 ohm preamplifier provides a minimum of 18 dB gain from 45 MHz to 1 GHz. (Power is supplied by the probe power output of the analyzer.)
- The 11909A low noise preamplifier provides a minimum of 32 dB gain from 9 kHz to 1 GHz and a typical noise figure of 1.8 dB.

GPIB Cable

The Agilent 10833 Series GPIB cables interconnect GPIB devices and are available in four different lengths (0.5 to 4 meters). GPIB cables are used to connect controllers to a signal analyzer.

USB/GPIB Cable

The Agilent 82357A USB/GPIB interface provides a direct connection from the USB port on your laptop or desktop PC to GPIB instruments. It comes with the SICL and VISA software for Windows® 2000/XP. Using VISA software, your existing GPIB programs work immediately, without modification. The 82357A is a standard Plug and Play device and you can interface with up to 14 GPIB instruments.

HP/Agilent 11970 Series Harmonic Mixers

The 11970 Series harmonic mixers are available to extend the frequency range of analyzers with Option AYZ (external mixing) up to

110 GHz. The following six models are available:

Table 1-1

HP/Agilent Model Number	Frequency Range
11970K	18.0 to 26.5 GHz
11970A	26.5 to 40.0 GHz
11970Q	33.0 to 50.0 GHz
11970U	40.0 to 60.0 GHz
11970V	50.0 to 75.0 GHz
11970W	75.0 to 110 GHz

HP/Agilent 11974 Series Preselected Millimeter Mixers

11974 Series preselected millimeter mixers are available to extend the frequency range of analyzers with Option AYZ (external mixing) up to 75 GHz. Preselection reduces mixer overload from broadband signals, reduces radiation of local oscillator harmonics back to the device under test, and reduces the level of image and multiple responses displayed. The following four models are available:

Table 1-2

HP/Agilent Model Number	Frequency Range
11974A	26.5 to 40.0 GHz
11974Q	33.0 to 50.0 GHz
11974U	40.0 to 60.0 GHz
11974V	50.0 to 75.0 GHz

RF and Transient Limiters

The Agilent 11867A and 11693A RF Limiters protect the analyzer input circuits from damage due to high power levels. The 11867A operates over a frequency range of dc to 1800 MHz and begins reflecting signal levels over 1 mW up to 10 W average power and 100 watts peak power. The 11693A microwave limiter (0.1 to 12.4 GHz, usable to 18 GHz) guards against input signals over 1 milliwatt up to 1 watt average power and 10 watts peak power.

The Agilent 11947A Transient Limiter protects the analyzer input circuits from damage due to signal transients. It specifically is needed for use with a line impedance stabilization network (LISN). It operates over a frequency range of 9 kHz to 200 MHz, with 10 dB of insertion loss.

Power Splitters

The Agilent 11667A/B/C power splitters are two-resister type splitters that provide excellent output SWR, at 50 Ω impedance. The tracking between the two output arms, over a broad frequency range, allows wideband measurements to be made with a minimum of uncertainty.

11667A: DC to 18 GHz
11667B: DC to 26.5 GHz
11667C: DC to 50 GHz

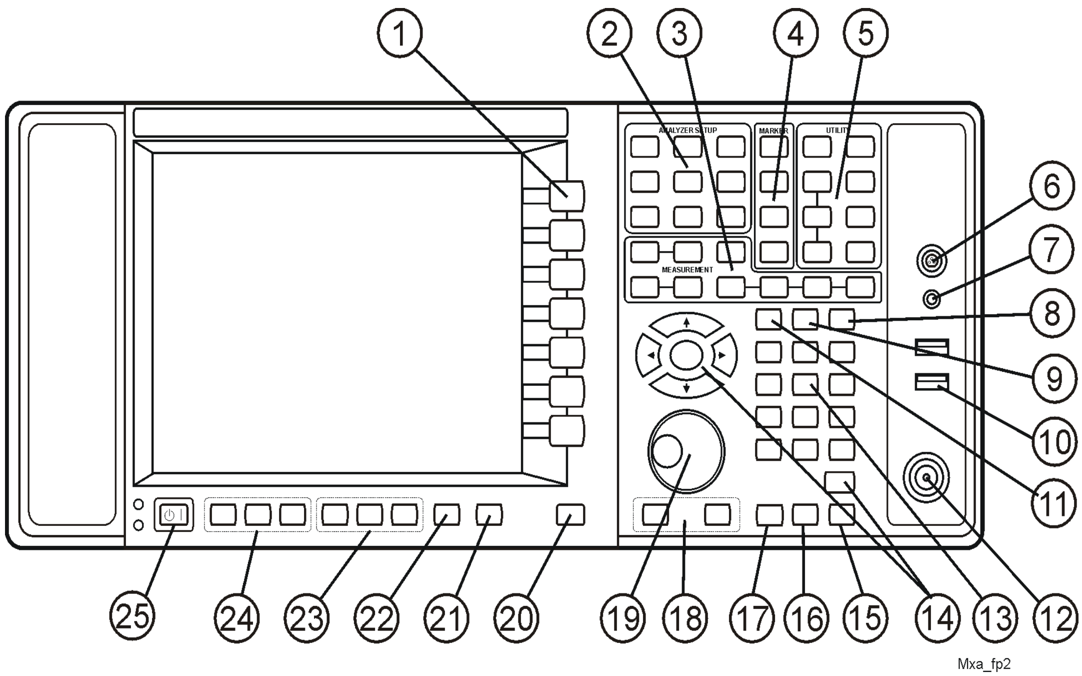
Static Safety Accessories

9300-1367	Wrist-strap, color black, stainless steel. Four adjustable links and a 7 mm post-type connection.
9300-0980	Wrist-strap cord 1.5 m (5 ft.)

2 Front and Rear Panel Features

Front Panel Features

Front-Panel Connectors and Keys



Item		Description
#	**Name**	
1	Menu Keys	Key labels appear to the left of the menu keys to identify the current function of each key. The displayed functions are dependent on the currently selected Mode and Measurement, and are directly related to the most recent key press.
2	Analyzer Setup Keys	These keys set the parameters used for making measurements in the current Mode and Measurement.
3	Measurement Keys	These keys select the Mode, and the Measurement within the mode. They also control the innitiation and frequency of measurement.
4	Marker Keys	Markers are often available for a measurement, to measure a very specific point/segment of data within the range of the current measurement data.
5	Utility Keys	These keys control system-wide functionaliy like: • instrument configuration information and I/O setup, • printer setup and printing, • file management, save and recall, • instrument instrument presets.

Item #	Item Name	Description
6	Probe Power	Supplies power for external high frequency probes and accessories.
7	Headphones Output	Headphones can be used to hear any available audio output.
8	Back Space Key	Press this key to delete the previous character when entering alphanumeric information. It also works as the Back key in Help and Explorer windows.
9	Delete Key	Press this key to delete files, or to perform other deletion tasks.
10	USB Connectors	Standard USB 2.0 ports, Type A. Connect to external peripherals such as a mouse, keyboard, DVD drive, or hard drive.
11	Local/Cancel/ (Esc) Key	If you are in remote operation, Local: • returns instrument control from remote back to local (the front panel). • turns the display on (if it was turned off for remote opperation). • can be used to clear errors. (Press the key once to return to local control, and a second time to clear error message line.) If you have not already pressed the units or Enter key, Cancel exits the currently selected function without changing its value. Esc works the same as it does on a pc keyboard. It: • exits Windows dialogs • resets input overloads • clears errors • aborts printing • cancels operations.
12	RF Input	Connector for inputing an external signal. Make sure that the total power of all signals at the analyzer input does *not* exceed +30 dBm (1 watt).
13	Numeric Keypad	Enters a specific numeric value for the current function.
14	Enter and Arrow Keys	The Enter key terminates data entry when either no unit of measure is needed, or you want to use the default unit. The arrow keys: • Increment and decrement the value of the current measurement selection. • Navigatie help topics. • Navigate, or make selections, within Windows dialogs. • Navigate within forms used for setting up measurements. • Navigate within tables. **NOTE** The arrow keys cannot be used to move a mouse pointer around on the display.
15	Menu/ (Alt) Key	Alt works the same as a pc keyboard. Use it to change control focus in Windows pull-down menus.
16	Ctrl Key	Ctrl works the same as a pc keyboard. Use it to navigate in Windows applications, or to select multiple items in lists.

Item #	Item Name	Description
17	Select / Space Key	Select is also the Space key and it has typical pc functionality. For example, in Windows dialogs, it selects files, checks and unchecks check boxes, and picks radio button choices. It opens a highlighted Help topic.
18	Tab Keys	Use these keys to move between fields in Windows dialogs.
19	Knob	Increments and decrements the value of the current active function.
20	Return Key	Exits the current menu and returns to the previous menu. Has typical pc functionality.
21	Full Screen Key	Pressing this key turns off the softkeys to maximize the graticule display area.
22	Help Key	Initiates a context-sensitive Help display for the current Mode. Once Help is accessed, pressing a front panel key brings up the help topic for that key function.
23	Speaker Control Keys	Enables you to increase or decrease the speaker volume, or mute it.
24	Window Control Keys	These keys select between single or multiple window displays. They zoom the current window to fill the data display, or change the currently selected window. They can be used to switch between the Help window navigation pane and the topic pane.
25	Power Standby On/Off	Turns the analyzer on. A green light indicates power on. A yellow light indicates standby mode. **NOTE** The front-panel switch is a standby switch, *not* a LINE switch (disconnecting device). The analyzer continues to draw power even when the line switch is in standby. The main power cord can be used as the system disconnecting device. It disconnects the mains circuits from the mains supply.

Overview of Key Types

The keys labeled **FREQ Channel**, **System**, and **Marker Function** are all examples of front-panel keys. Most of the dark or light gray keys access menus of functions that are displayed along the right side of the display. These displayed key labels are next to a column of keys called menu keys.

Menu keys list functions based on which front-panel key was pressed last. These functions are also dependant on the current selection of measurement application (**Mode**) and measurement (**Meas**).

If a menu key function's numeric value can be changed, it is called an active function. The function label of the active function is highlighted after that key has been selected. The displayed value indicates that the function is selected and its value can now be changed using any of the data entry controls.

Some menu keys have multiple choices on their label like On/Off or Auto/Man. The different choices are selected by pressing the key multiple times. Take an Auto/Man type of key as an example. To select the function, press the menu key and notice that Auto is underlined and the key becomes highlighted. To change the function to manual, press the key again so that Man is underlined. If there are more than two settings on the key, keep pressing it until the desired selection is underlined.

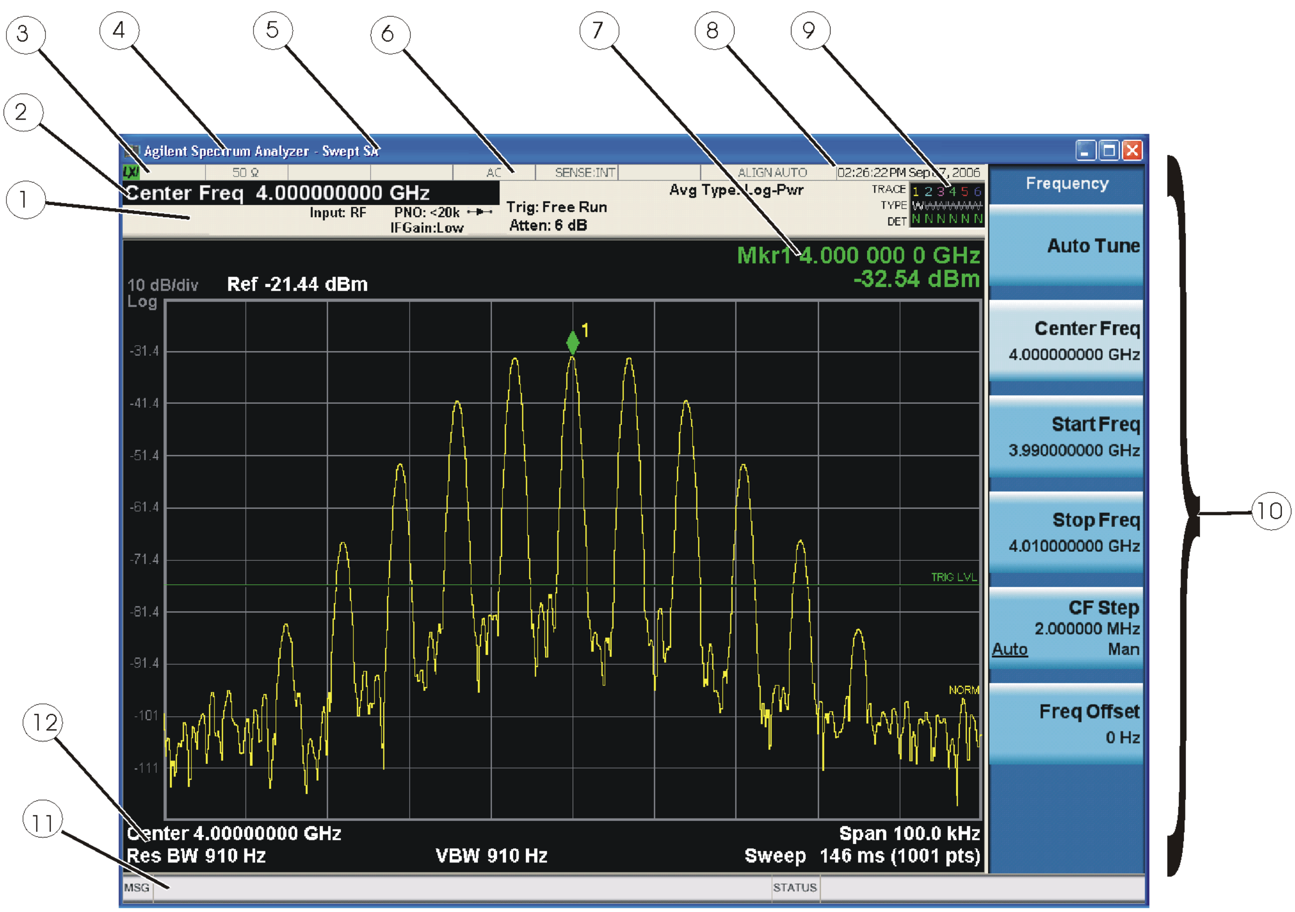
When a menu first appears, one key label will be highlighted to show which key is the default selection. Some of the menu keys are grouped together by a yellow bar running behind the keys near the left side. When you press a key within the yellow bar region the highlight will move to that key to show it has been selected. The keys that are linked by the yellow bar are related functions, and only one of them can be selected at any one time. If the current menu is two pages long, the yellow bar could include keys on the second page of keys.

In some key menus, a key label will be highlighted to show which key has been selected from multiple available choices. And the menu is immediately exited when you press one of the other keys.

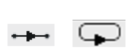
If a displayed key label shows a small solid-black arrow tip pointing to the right, it indicates that additional key menus are available. If the arrow tip is not filled in solid then pressing the key the first time selects that function. Now the arrow is solid and pressing it again will bring up an additional menu of settings.

Display Annotations

The following graphic is an example of an N9060A Spectrum Analyzer display. Your instrument display may look different.

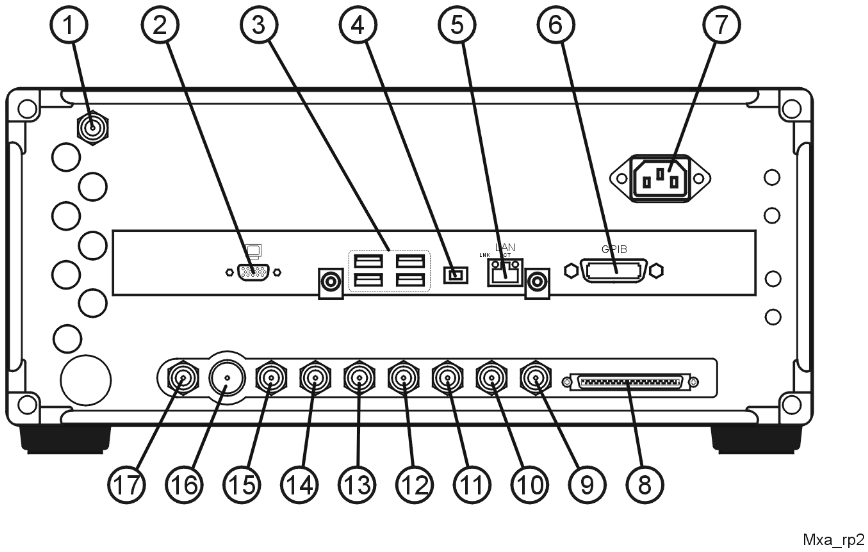


Displayannot

Item	Description	Function Keys
1	Measurement bar - Shows general measurement settings and information.  Indicates single/continuous measurement. Some measurements include limits that the data is tested against. A Pass/Fail indication may be shown in the lower left of the measurement bar.	All the keys in the Analyzer Setup part of the front panel.
2	Active Function (measurement bar) - when the current active function has a settable numeric value, it is shown here.	Currently selected front panel key.

Item	Description	Function Keys
3	Banner - shows the name of the selected measurement application and the measurement that is currently running.	**Mode**, **Meas**
4	Measurement title (banner) - shows title information for the current Measurement, or a title that you created for the measurement.	**Meas** **View/Display**, **Display**, **Title**
5	Settings panel - displays system information that is not specific to any one application. • Input/Output status - green LXI indicates the LAN is connected. RLTS indicate Remote, Listen, Talk, SRQ • Input impedance and coupling • Selection of external frequency reference • Setting of automatic internal alignment routine	**Local** and **System, I/O Config** **Input/Output**, **Amplitude**, **System** and others
6	Active marker frequency, amplitude or function value	**Marker**
7	Settings panel - time and date display.	**System**, **Control Panel. . .**
8	Trace/Detector panel (measurement bar) **Type:** / **Detector:** W - clear/write / N - normal detection (rise and fall) A - trace average / S - sample detection M - max hold / P - positive peak detection m - min hold / p - negative peak detection A - average detection White = update on / f - math function Gray = update off Srikethrough "~~W~~" means display off	**Trace/Detector**  Trace: Identifies traces by color Type: Shows trace mode setting, trace display on/off, and trace update on/off Det: Shows detector selection or math function applied to the trace
9	Key labels that change based on the most recent key press.	Softkeys
10	Displays information, warning and error messages. Message area - single events, Status area - conditions	
11	Measurement settings for the data currently being displayed in the graticule area. In the example above: center frequency, resolution bandwidth, video bandwidth, frequency span, sweep time and number of sweep points.	Keys in the Analyzer Setup part of the front panel.

Rear-Panel Features



Item		Description
#	Name	
1	EXT REF IN	Input for a 1 to 50 MHz external frequency reference signal.
2	MONITOR	Allows connection of an external VGA monitor.
3	USB Connectors	Standard USB 2.0 ports, Type A. Connect to external peripherals such as a mouse, keyboard, printer, DVD drive, or hard drive.
4	USB Connector	USB 2.0 port, Type B. USB TMC (test and measurement class) connects to an external pc controller to control the instrument and for data transfers over a 480 Mbps link.
5	LAN	A TCP/IP Interface that is used for remote analyzer operation.
6	GPIB	A General Purpose Interface Bus (GPIB, IEEE 488.1) connection that can be used for remote analyzer operation.
7	Line power input	The AC power connection. See the product specifications for more details.
8	Digital Bus	Reserved for future use.
9	Analog Out	Reserved for future use.
10	TRIGGER 2 OUT	A trigger output used to synchronize other test equipment with the analyzer. Configurable from the Input/Output keys.
11	TRIGGER 1 OUT	A trigger output used to synchronize other test equipment with the analyzer. Configurable from the Input/Output keys.
12	Sync	Reserved for future use.

Item #	Item Name	Description
13	TRIGGER 2 IN	Allows external triggering of measurements.
14	TRIGGER 1 IN	Allows external triggering of measurements.
15	Noise Source Drive +28 V (Pulsed)	Reserved for future use.
16	SNS Series Noise Source	Reserved for future use.
17	10 MHz OUT	An output of the analyzer’s internal 10 MHz frequency reference signal. It is used to lock the frequency reference of other test equipment to the analyzer.

Front and Rear Panel Symbols

This symbol is used to indicate power ON (green LED).

This symbol is used to indicate power STANDBY mode (yellow LED).

This symbol indicates the input power required is AC.

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

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

N10149

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

ICES/NMB-001
ISM GRP.1 CLASS A

This is a marking of a product in compliance with the Canadian Interference-Causing Equipment Standard (ICES-001).

This is also a symbol of an Industrial Scientific and Medical Group 1 Class A product (CISPR 11, Clause 4).

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

This symbol indicates separate collection for electrical and electronic equipment mandated under EU law as of August 13, 2005. All electric and electronic equipment are required to be separated from normal waste for disposal (Reference WEEE Directive 2002/96/EC).

To return unwanted products, contact your local Agilent office, or see http://www.agilent.com/environment/product/ for more information.

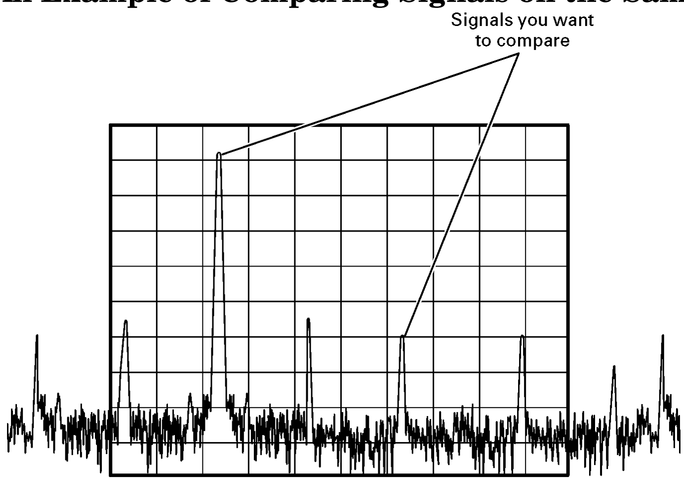
3 Measuring Multiple Signals

Comparing Signals on the Same Screen Using Marker Delta

Using the analyzer, you can easily compare frequency and amplitude differences between signals, such as radio or television signal spectra. The analyzer delta marker function lets you compare two signals when both appear on the screen at one time.

In this procedure, the analyzer 10 MHz signal is used to measure frequency and amplitude differences between two signals on the same screen. Delta marker is used to demonstrate this comparison.

Figure 3-1 **An Example of Comparing Signals on the Same Screen**



Step 1. Connect the `10 MHz OUT` from the rear panel to the front panel RF input.

Step 2. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 3. Preset the spectrum analyzer mode:

Press **Mode Preset**.

Step 4. Set the analyzer center frequency, span and reference level to view the 10 MHz signal and its harmonics up to 50 MHz:

Press **FREQ Channel**, **Center Freq**, 30, **MHz**.
Press **SPAN X Scale**, **Span**, 50, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 10, **dBm**.

Step 5. Place a marker at the highest peak on the display (10 MHz):

Press **Peak Search**.

The **Next Pk Right** and **Next Pk Left** softkeys are available to move the marker from peak to peak. The marker should be on the 10 MHz reference signal:

Step 6. Anchor the first marker and activate a second delta marker:

Press **Marker, Delta**.

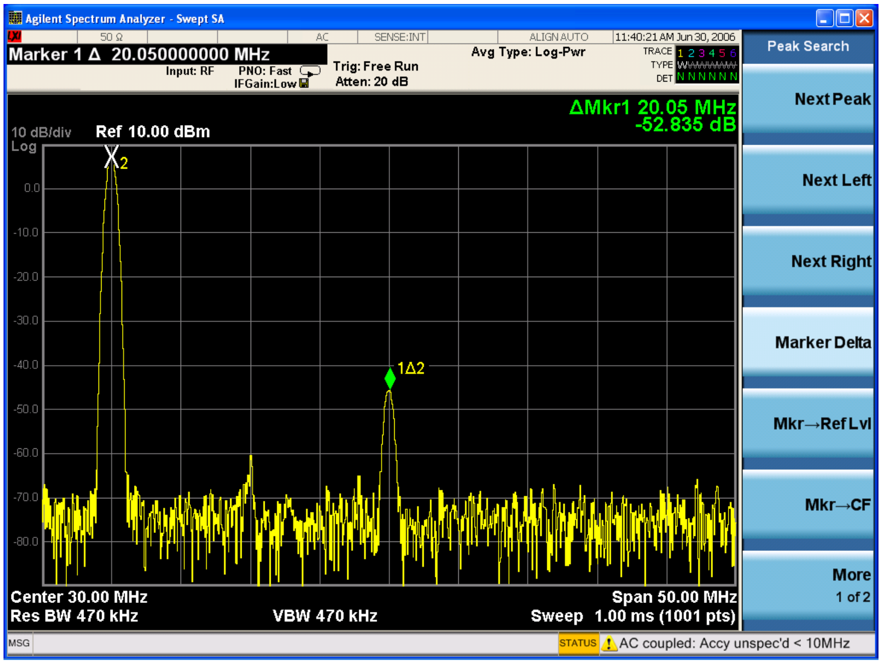
The symbol for the first marker is changed from a diamond to a cross (X) with a label that now reads 2, indicating that it is the fixed marker (reference point). The second marker symbol is a diamond labeled 1Δ2, indicating it is the delta marker. When you first press the Delta key, both markers are at the same frequency with the symbols superimposed over each other. It is not until you move the delta marker to a new frequency that the different marker symbols are easy to discern.

Step 7. Move the delta marker to another signal peak using the front-panel knob or by using the **Peak Search** key:

Press **Peak Search, Next Peak** or
Press **Peak Search**, **Next Pk Right** or **Next Pk Left**.

The amplitude and frequency *difference* between the markers is displayed in the marker results block of the screen. Refer to the upper right portion of the screen. Refer to Figure 3-2.

Figure 3-2 **Using the Delta Marker Function**



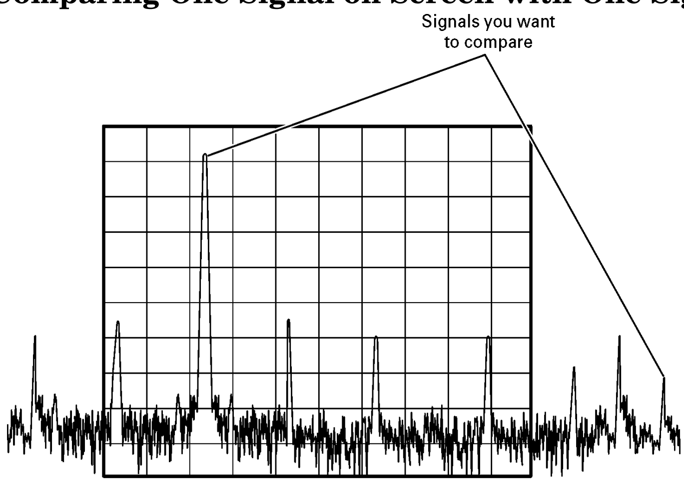
NOTE The frequency resolution of the marker readings can be increased by turning on the marker count function.

Comparing Signals not on the Same Screen Using Marker Delta

Measure the frequency and amplitude difference between two signals that do not appear on the screen at one time. (This technique is useful for harmonic distortion tests when narrow span and narrow bandwidth are necessary to measure the low level harmonics.)

In this procedure, the analyzer 10 MHz signal is used to measure frequency and amplitude differences between one signal on screen and one signal off screen. Delta marker is used to demonstrate this comparison.

Figure 3-3 Comparing One Signal on Screen with One Signal Off Screen



Step 1. Connect the `10 MHz OUT` from the rear panel to the front panel RF input.

Step 2. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 3. Preset the analyzer:

Press **Mode Preset**.

Step 4. Set the analyzer center frequency, span and reference level to view the 10 MHz signal and its harmonics up to 50 MHz:

Press **FREQ Channel**, **Center Freq**, 10, **MHz**.
Press **SPAN X Scale**, **Span**, 5, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 10, **dBm**.

Step 5. Place a marker on the 10 MHz peak and then set the center frequency step size equal to the marker frequency (10 MHz):

Press **Peak Search**.
Press **Marker** →, **Mkr** → **CF Step**.

Step 6. Activate the marker delta function:

Press **Marker, Delta**.

Step 7. Increase the center frequency by 10 MHz:

Press **FREQ Channel, Center Freq**, ↑.

The first marker and delta markers move to the left edge of the screen, at the amplitude of the first signal peak.

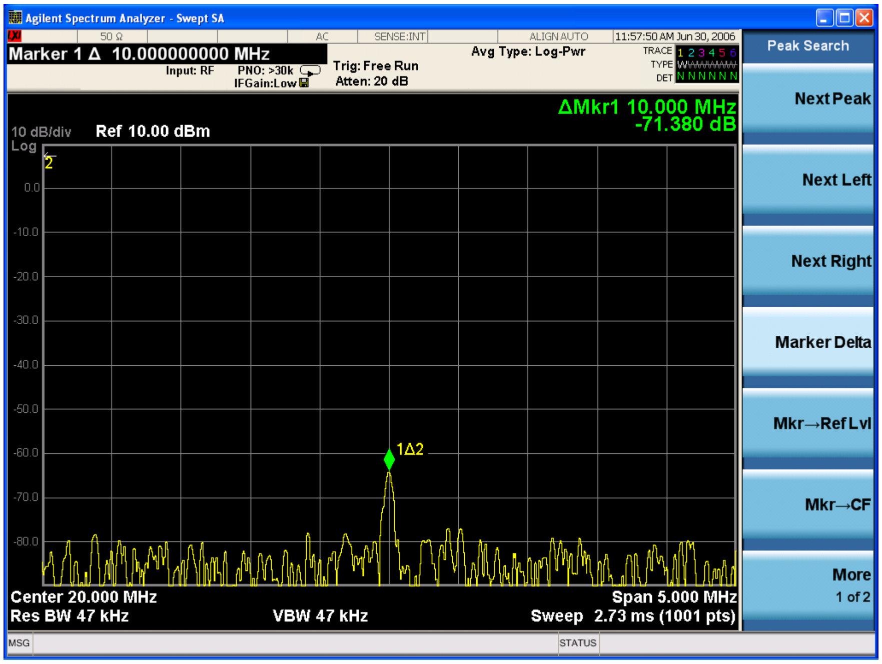
Step 8. Move the delta marker to the new center frequency:

Press **Peak Search**.

Figure 3-4 shows the reference annotation for the first marker (×2) at the left side of the display, indicating that the 10 MHz reference signal is at a lower frequency than the frequency range currently displayed.

The delta marker (1Δ2) appears on the peak of the 20 MHz component. The delta marker results block displays the amplitude and frequency difference between the 10 and 20 MHz signal peaks.

Figure 3-4 Delta Marker with Reference Signal Off-Screen



Step 9. Turn the markers off:

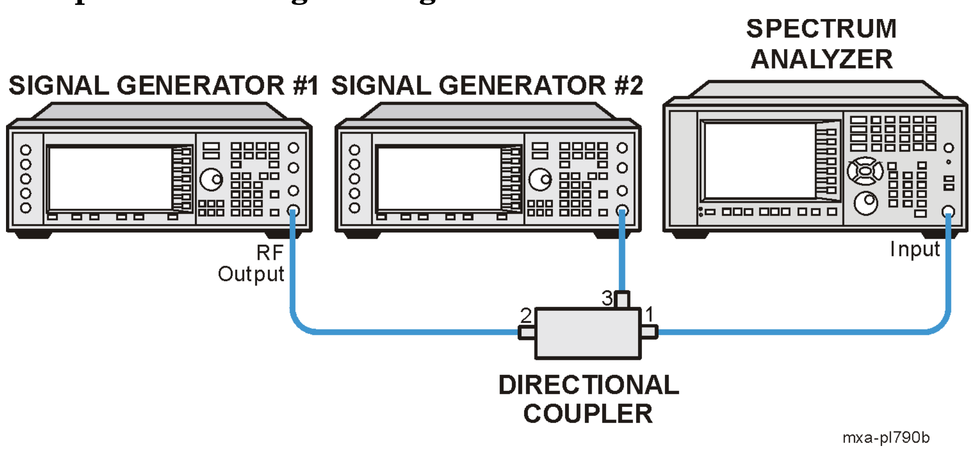
Press **Marker, Off**.

Resolving Signals of Equal Amplitude

In this procedure a decrease in resolution bandwidth is used in combination with a decrease in video bandwidth to resolve two signals of equal amplitude with a frequency separation of 100 kHz. Notice that the final RBW selection to resolve the signals is the same width as the signal separation while the VBW is slightly narrower than the RBW.

Step 1. Connect two sources to the analyzer RF INPUT as shown in Figure 3-5.

Figure 3-5 **Setup for Obtaining Two Signals**



Step 2. Setup the signal sources as follows:

Set one source to 300 MHz.
Set the frequency of the other source to 300.1 MHz.
Set both source amplitudes to –20 dBm. The amplitude of both signals should be approximately –20 dBm at the output of the bridge/directional coupler.

Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

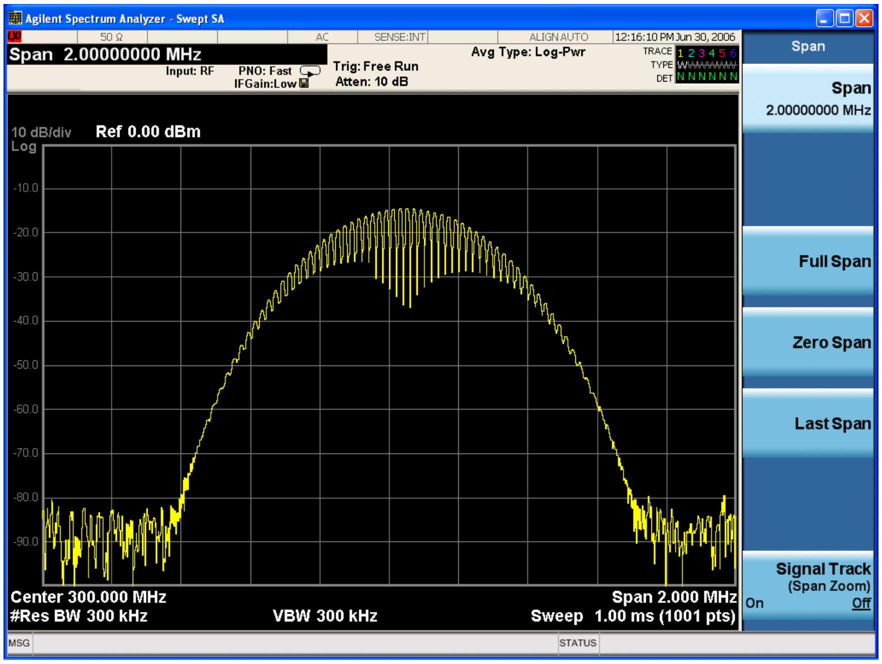
Press **Mode Preset**.

Step 5. Setup the analyzer to view the signals:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **BW**, **Res BW**, 300, **kHz**.
Press **SPAN X Scale**, **Span**, 2, **MHz**.

A single signal peak is visible. See Figure 3-6.

Figure 3-6 Unresolved Signals of Equal Amplitude

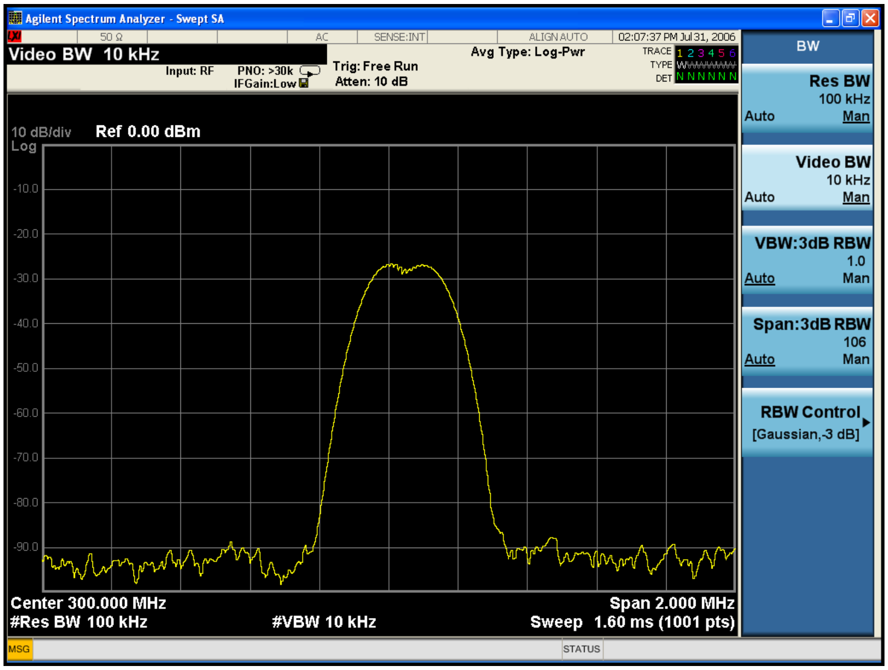


Step 6. Change the resolution bandwidth (RBW) to 100 kHz so that the RBW setting is less than or equal to the frequency separation of the two signals and decrease the video bandwidth to 10 kHz:

Press **BW, Res BW**, 100, **kHz**.
Press **Video BW**, 10, **kHz**

Notice that the peak of the signal has become two peaks separated by a 2.5 dB dip indicating that two signals may be present. Refer to Figure 3-7

Figure 3-7 Unresolved Signals of Equal Amplitude

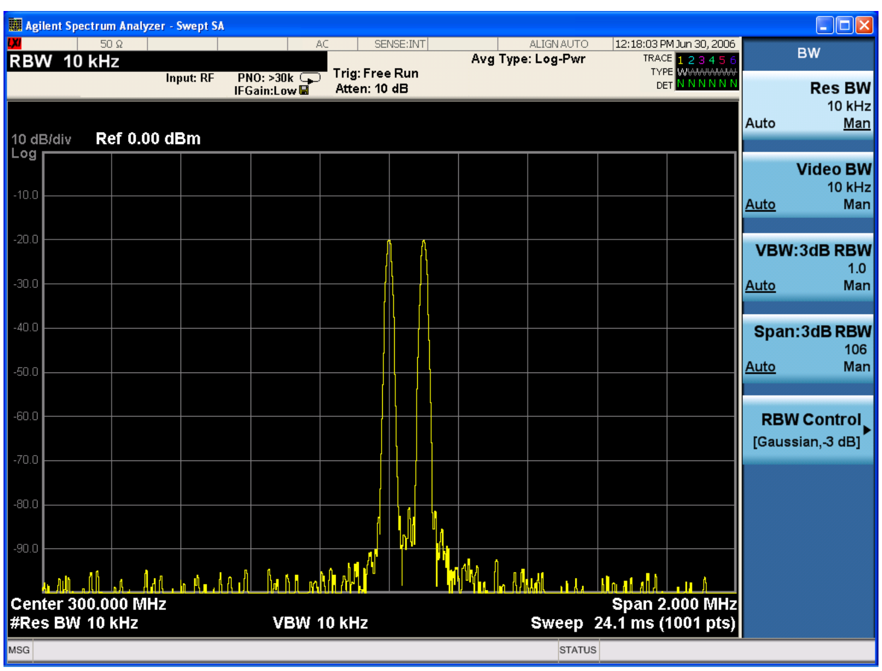


Step 7. Decrease the resolution bandwidth (RBW) to 10 kHz:

Press **BW, Res BW**, 10, **kHz**.

Two signals are now visible as shown in Figure 3-8. You can use the front-panel knob or step keys to further reduce the resolution bandwidth and better resolve the signals.

Figure 3-8 Resolving Signals of Equal Amplitude



As the resolution bandwidth is decreased, resolution of the individual signals is improved and the sweep time is increased. For fastest measurement times, use the widest possible resolution bandwidth. Under mode preset conditions, the resolution bandwidth is “coupled” (or linked) to the span.

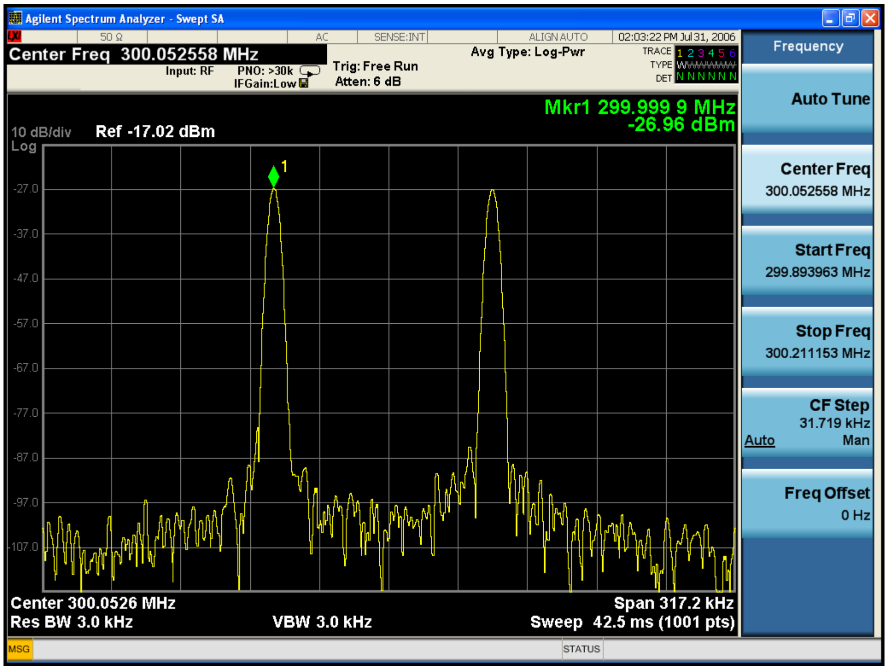
Since the resolution bandwidth has been changed from the coupled value, a # mark appears next to `Res BW` in the lower-left corner of the screen, indicating that the resolution bandwidth is uncoupled. (For more information on coupling, refer to the **Auto Couple** key description in the *Agilent Technologies User’s and Programmer’s Reference N9060A Spectrum Analyzer Mode*.)

NOTE An alternative method for resolving two equal amplitude signals is to use the Auto Tune Function as follows:

Press Mode Preset.
Press Freq Channel, Auto Tune

The two signals are fully resolved with a marker placed on the first peak. refer to Figure 3-9.

Figure 3-9 Resolving Signals of Equal Amplitude

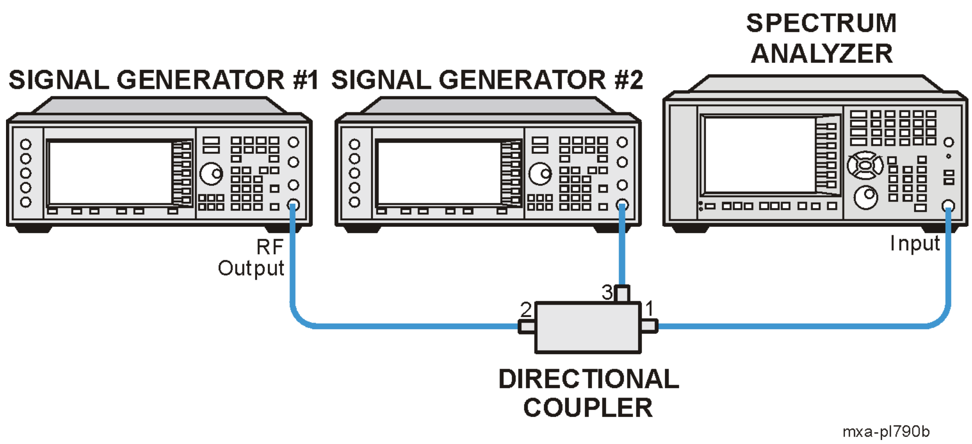


Resolving Small Signals Hidden by Large Signals

This procedure uses narrow resolution bandwidths to resolve two input signals with a frequency separation of 155 kHz and an amplitude difference of 60 dB.

Step 1. Connect two sources to the analyzer RF INPUT as shown in Figure 3-10.

Figure 3-10 **Setup for Obtaining Two Signals**



Step 2. Setup the signal sources as follows:

Set one source to 300 MHz at –10 dBm.
Set the second source to 300.05 MHz, so that the signal is 50 kHz higher in frequency than the first signal.
Set the amplitude of the second source signal to –70 dBm (60 dB below the first signal).

Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer as follows:

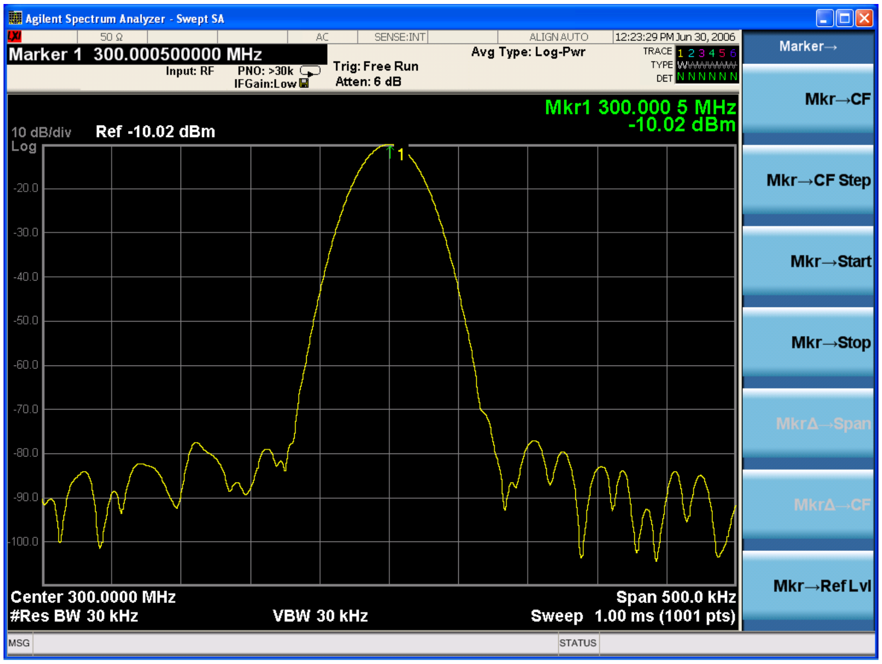
Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **BW**, 30, **kHz**.
Press **SPAN X Scale**, **Span**, 500, **kHz**.

Step 6. Set the 300 MHz signal to the reference level:

Press **Peak Search**, **Mkr** → **Ref Lvl**.

NOTE The MXA Signal Analyzer 30 kHz filter shape factor of 4.1:1 has a bandwidth of 123 kHz at the 60 dB point. The half-bandwidth, or 61.5 kHz, is NOT narrower than the frequency separation of 50 kHz, so the input signals can not be resolved. Refer to Figure 3-11.

Figure 3-11 Signal Resolution with a 30 kHz RBW

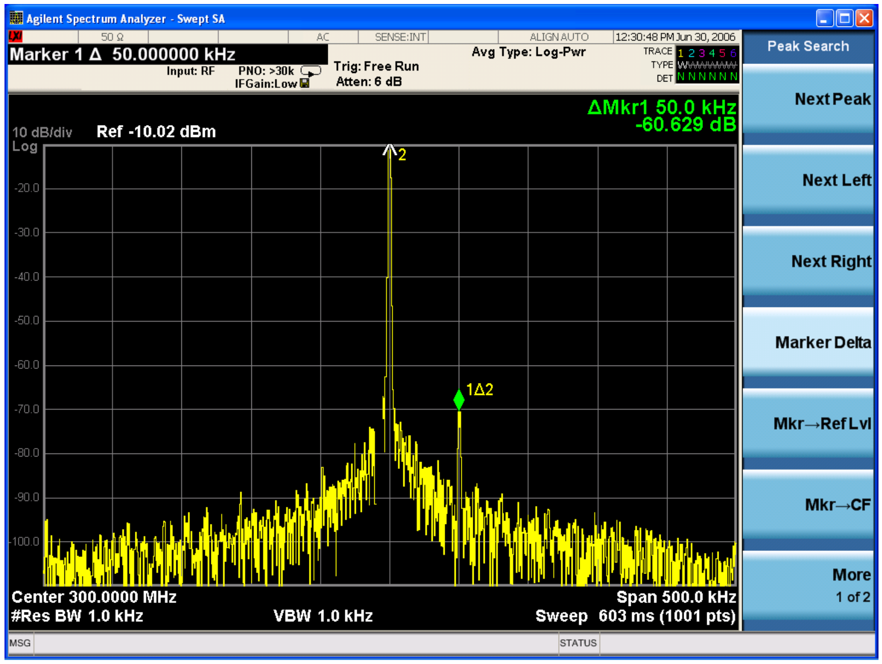


Step 7. Reduce the resolution bandwidth filter to view the smaller hidden signal. Place a delta marker on the smaller signal:

Press **BW**, 1, **kHz**.
Press **Peak Search**, **Marker Delta**, 50, **kHz**.

NOTE The MXA Signal Analyzer 1 kHz filter shape factor of 4.1:1 has a bandwidth of 4.1 kHz at the 60 dB point. The half-bandwidth, or 2.05 kHz, is narrower than 50 kHz, so the input signals can be resolved. Refer to Figure 3-12.

Figure 3-12 Signal Resolution with a 1 kHz RBW



Decreasing the Frequency Span Around the Signal

Using the analyzer signal track function, you can quickly decrease the span while keeping the signal at center frequency. This is a fast way to take a closer look at the area around the signal to identify signals that would otherwise not be resolved.

This procedure uses signal tracking with span zoom to view the analyzer 50 MHz reference signal in a 200 kHz span.

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Enable the internal 50 MHz amplitude reference signal of the analyzer as follows:

Press **Input/Output**, **RF Calibrator**, **50 MHz**.

Step 4. Set the start frequency to 20 MHz and the stop frequency to 1 GHz:

Press **FREQ Channel**, **Start Freq**, 20, **MHz**.
Press **FREQ Channel**, **Stop Freq**, 1, **GHz**.

Step 5. Turn on the signal tracking function to place a marker at the peak and move the signal to the center of the screen (if it is not already positioned there) and initiate Span Zoom:

Press **Span X Scale**, **Signal Track (Span Zoom)** (`On`).

See the left-side of figure Figure 3-13.

Step 6. Set the 50 MHz calibration signal to the reference level:

Press **Mkr** →, **Mkr** →Ref Lvl.

NOTE Because the signal track function automatically maintains the signal at the center of the screen, you can reduce the span quickly for a closer look. If the signal drifts off of the screen as you decrease the span, use a wider frequency span.

Step 7. Reduce span and resolution bandwidth to further zoom in on the marked signal:

Press **SPAN X Scale**, **Span**, **200**, **kHz**.

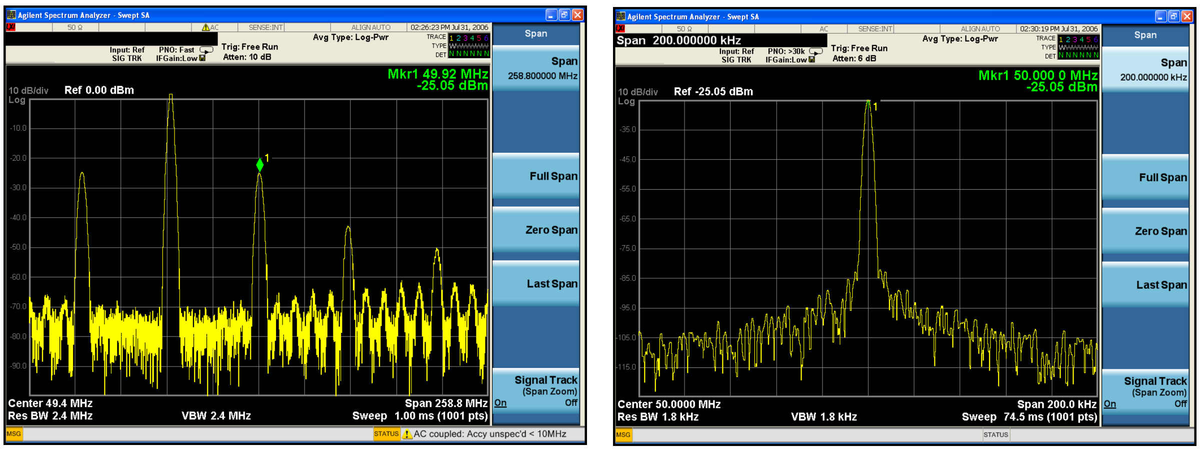
See the right-side of figure Figure 3-13.

NOTE If the span change is large enough, the span decreases in steps as automatic zoom is completed. See Figure 3-13 on the right side. You can also use the front-panel knob or step keys to decrease the span and resolution bandwidth values.

Step 8. Turn off signal tracking:

Press **SPAN X Scale**, **Signal Track** (`Off`).

Figure 3-13 Signal Tracking



LEFT: Signal tracking on before span decrease
RIGHT: After zooming in on the signal

Easily Measure Varying Levels of Modulated Power Compared to a Reference

This section demonstrates a method to measure the complex modulated power of a reference device or setup and then compare the result of adjustments and changes to that or other devices.

The Delta Band/Interval Power Marker function will be used to capture the simulated signal power of a reference device or setup and then compare the resulting power level due to adjustments or DUT changes.

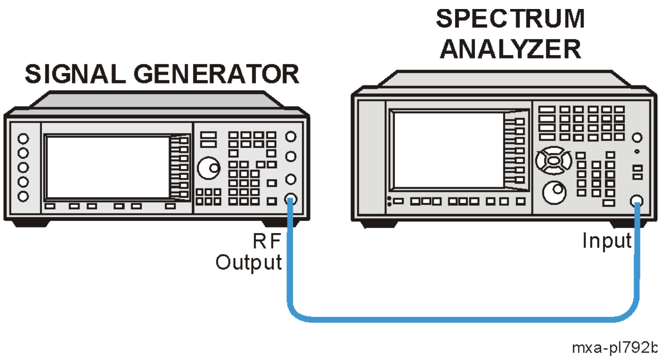
An important key to making accurate Band Power Marker measurements is to insure that the Average Type under the Meas Setup key is set to “Auto”.

Step 1. Setup the signal source as follows:

Setup a 4-carrier W-CDMA signal
Set the source frequency to 1.96 GHz.
Set the source amplitude to –10 dBm

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 3-14.

Figure 3-14 **Setup for Signal-to-Noise Measurement**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Tune to the W-CDMA signal and set the MXA reference level:

Press **FREQ Channel**, **Auto Tune**
Press **AMPTD Y Scale**, **Ref Level**, 0, **dBm**

Step 6. Enable trace averaging and the Band/Interval Power Marker function for measuring the total power of the reference 4-carrier W-CDMA signal.

Press **Trace/Detector**, **Select Trace**, **Trace 1**, **Trace Average**
Press **Marker Function**, **Band/Interval Power**

Step 7. Center the frequency of the Band/Interval Power marker on the 4-carrier reference signal envelope:

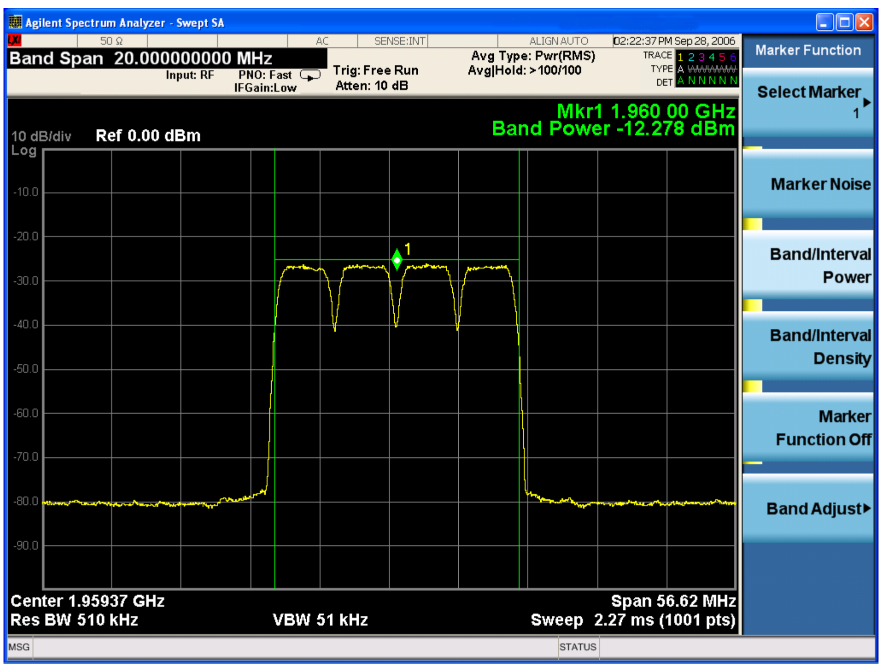
Press **Select Marker**, **Marker 1**, 1.96, **GHz**

Step 8. Adjust the width (or span) of the Band/Interval Power marker to encompass the entire 4-carrier W-CDMA reference signal (refer to Figure 3-15):

Note the green vertical lines of Marker 1 representing the span of signals included in the Band/Interval Power measurement and the carrier power indicated in Markers Result Block.

Press **Marker Function**, **Band Adjust**, **Band/Interval Span**, 20, **MHz**

Figure 3-15 Measured Power of Reference 4-Carrier W-CDMA Signal using Band/Interval Power Marker



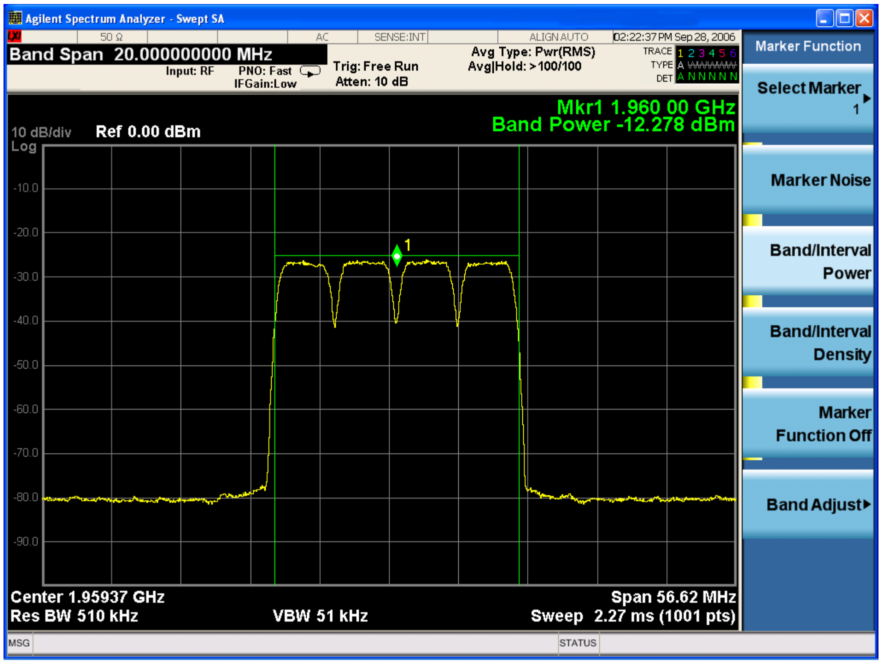
Step 9. Enable the Delta Band Power Marker functionality which will change the reference Band Power Marker into a fixed power value (labeled X_2) and initiate a second Band Power Marker (labeled 1Δ2) to measure any changes in power levels relative to the reference Band Power Marker X_2.

Press **Marker**, **Select Marker**, **Marker 1**, **Delta**

Step 10. Simulate a varying power level resulting from either adjustments, changes to the reference DUT or a different DUT by lowering the signal source power. Note the Delta Band Power Marker value displayed in the Marker Result Block showing the 10 dB difference between the modulated power of the reference and the changed power level (refer to Figure 3-16).

Set the source amplitude to –20 dBm

Figure 3-16 Delta Band Power Markers Displaying Lower Modulated Power Level Compared to a Reference



4 Measuring a Low–Level Signal

Reducing Input Attenuation

The ability to measure a low-level signal is limited by internally generated noise in the MXA Signal Analyzer. The measurement setup can be changed in several ways to improve the analyzer sensitivity.

The input attenuator affects the level of a signal passing through the instrument. If a signal is very close to the noise floor, reducing input attenuation can bring the signal out of the noise.

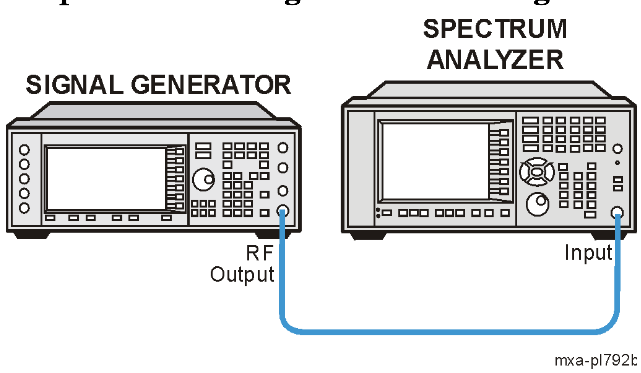
CAUTION Ensure that the total power of all input signals at the analyzer RF input does not exceed +30 dBm (1 watt).

Step 1. Setup the signal source as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –80 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 4-1.

Figure 4-1 **Setup for Measuring a Low-Level Signal**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the center frequency, span and reference level:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 5, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 40, **–dBm**.

Step 6. Move the desired peak (in this example, 300 MHz) to the center of the display:

Press **Peak Search**, **Marker →**, **Mkr → CF**.

Step 7. Reduce the span to 1 MHz (as shown in Figure 4-2) and if necessary re-center the peak:

Press **Span**, 1, **MHz**.

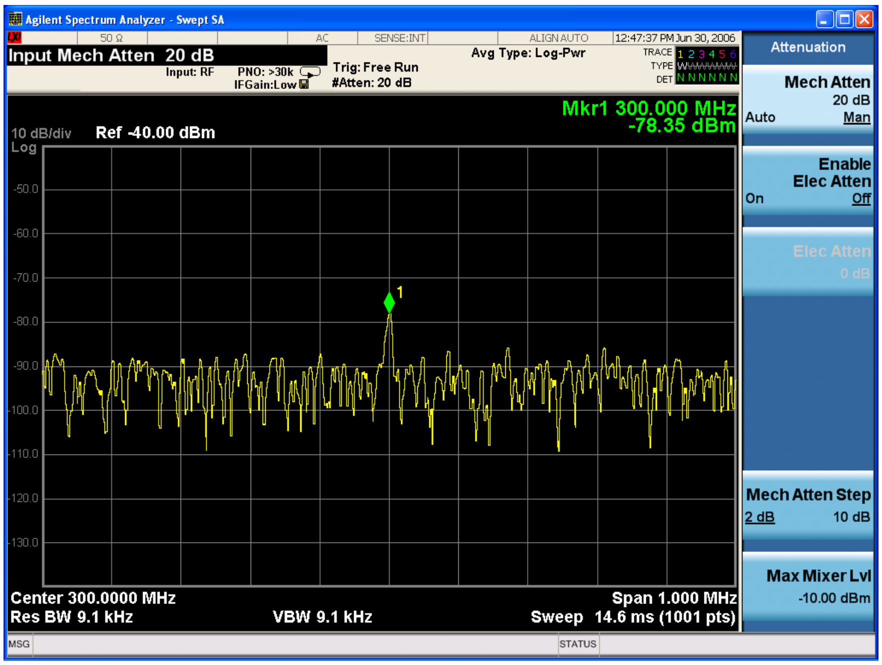
Step 8. Set the attenuation to 20 dB:

Press **AMPTD Y Scale**, **Attenuation**, **Mech Atten** (Man), 20, **dB**.

Note that increasing the attenuation moves the noise floor closer to the signal level.

A “#” mark appears next to the `Atten` annotation at the top of the display, indicating that the attenuation is no longer coupled to other analyzer settings.

Figure 4-2 Measuring a Low-Level Signal Mechanical Attenuation

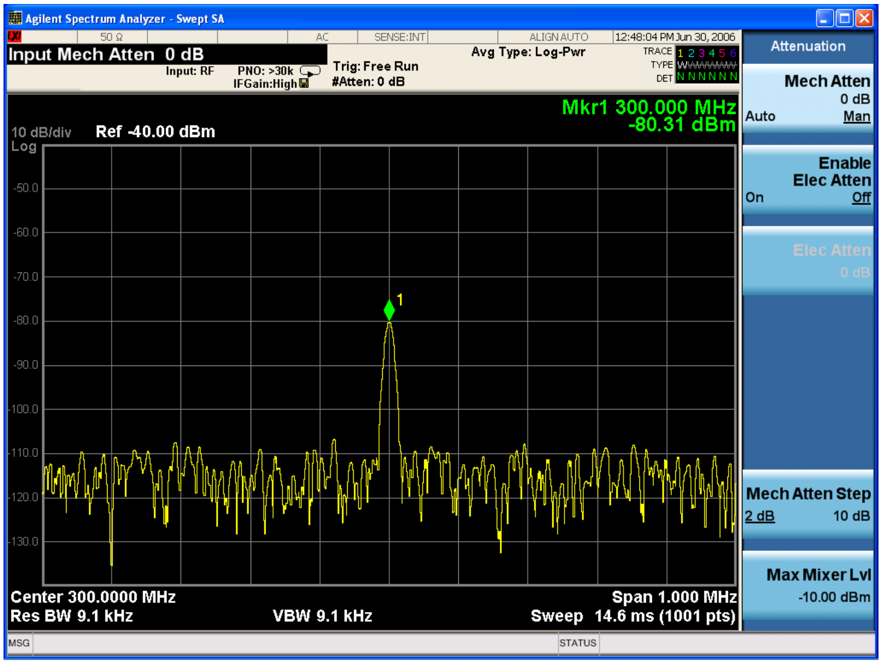


Step 9. To see the signal more clearly, set the attenuation to 0 dB:

Press **AMPTD Y Scale**, **Attenuation**, **Mech Atten** (Man), 0, **dB**.

See Figure 4-3 shows 0 dB input attenuation.

Figure 4-3 Measuring a Low-Level Signal Using 0 dB Attenuation



CAUTION When you finish this example, increase the attenuation to protect the analyzer's RF input:

Press **AMPTD Y Scale**, **Attenuation**, **Mech Atten** (Auto) or press **Auto Couple**.

Decreasing the Resolution Bandwidth

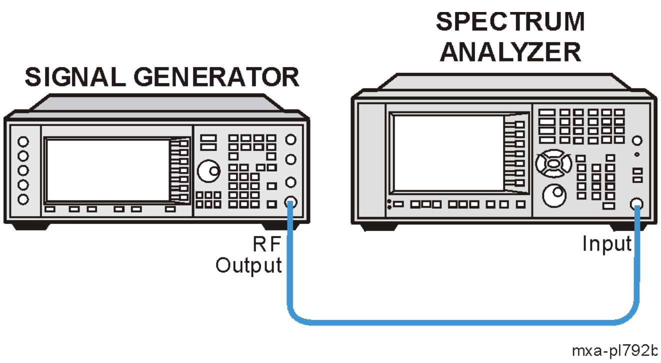
Resolution bandwidth settings affect the level of internal noise without affecting the level of continuous wave (CW) signals. Decreasing the RBW by a decade reduces the noise floor by 10 dB.

Step 1. Setup the signal sources as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –80 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 4-4.

Figure 4-4 **Setup for Measuring a Low-Level Signal**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

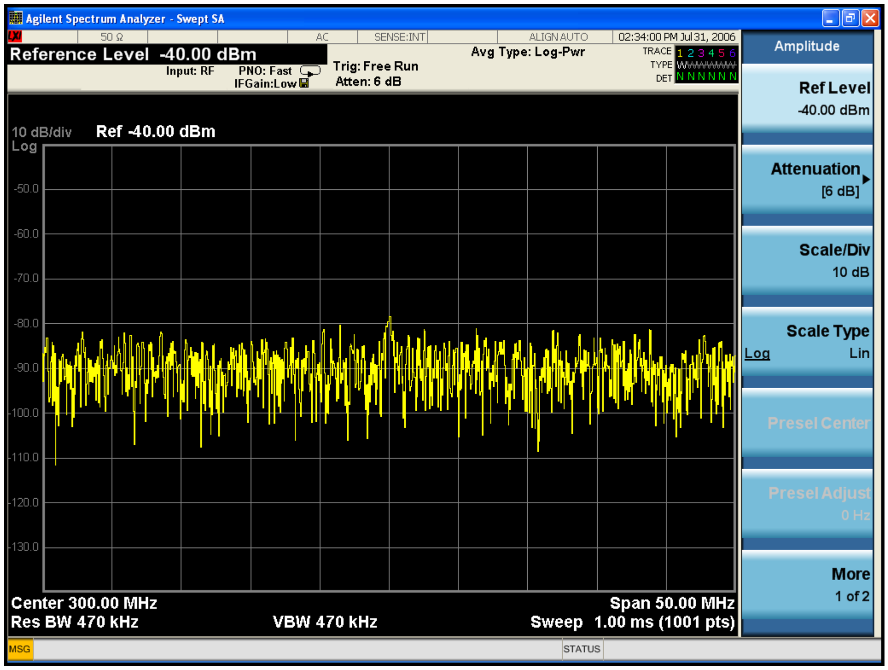
Press **Mode Preset**.

Step 5. Set the center frequency, span and reference level:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 50, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 40, **–dBm**.

Refer to Figure 4-5.

Figure 4-5 Default Resolution Bandwidth

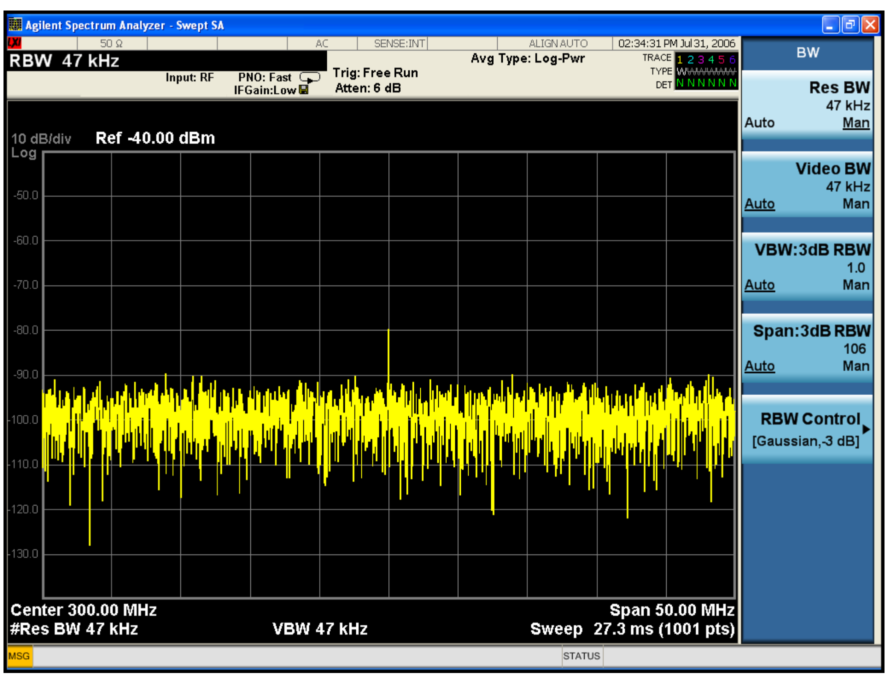


Step 6. Decrease the resolution bandwidth:

Press **BW**, 47, **kHz**.

The low-level signal appears more clearly because the noise level is reduced (refer to Figure 4-6).

Figure 4-6 Decreasing Resolution Bandwidth



A "#" mark appears next to the `Res BW` annotation in the lower left corner of the screen, indicating that the resolution bandwidth is uncoupled.

RBW Selections

You can use the step keys to change the RBW in a 1–3–10 sequence.

All MXA Signal Analyzer RBWs are digital and have a selectivity ratio of 4.1:1. Choosing the next lower RBW (in a 1–3–10 sequence) for better sensitivity increases the sweep time by about 10:1 for swept measurements, and about 3:1 for FFT measurements (within the limits of RBW). Using the knob or keypad, you can select RBWs from 1 Hz to 3 MHz in approximately 10% increments, plus 4, 5, 6 and 8 MHz. This enables you to make the trade off between sweep time and sensitivity with finer resolution.

Using the Average Detector and Increased Sweep Time

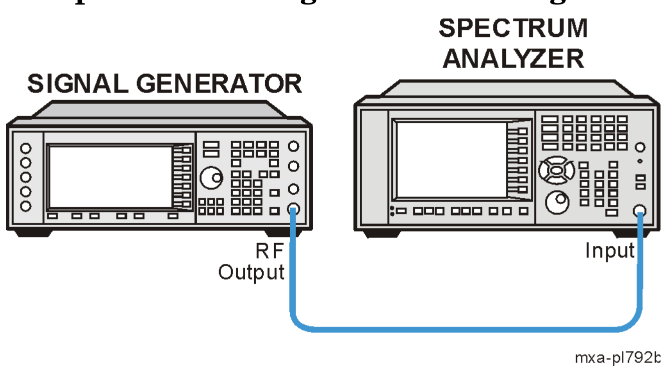
When the analyzer's noise masks low-level signals, changing to the average detector and increasing the sweep time smooths the noise and improves the signal's visibility. Slower sweeps are required to average more noise variations.

Step 1. Setup the signal sources as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –80 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 4-7.

Figure 4-7 Setup for Measuring a Low-Level Signal



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the center frequency, span and reference level:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 5, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 40, **–dBm**.

Step 6. Select the average detector:

Press Trace/Detector, **More**, **Detector**, **Average** (Log/RMS/V).

The number 1 (Trace 1 indicator) in the Trace/Detector panel (in the upper right-hand corner of the display) changes from green to white, indicating that the detector has been chosen manually. In addition, the letter in the Det row has ben set to "A" indicating that the Average detector has been selected. (see Figure 4-8).

Step 7. Increase the sweep time to 100 ms:

Press **Sweep/Control**, **Sweep Time** (Man), 100, **ms**.

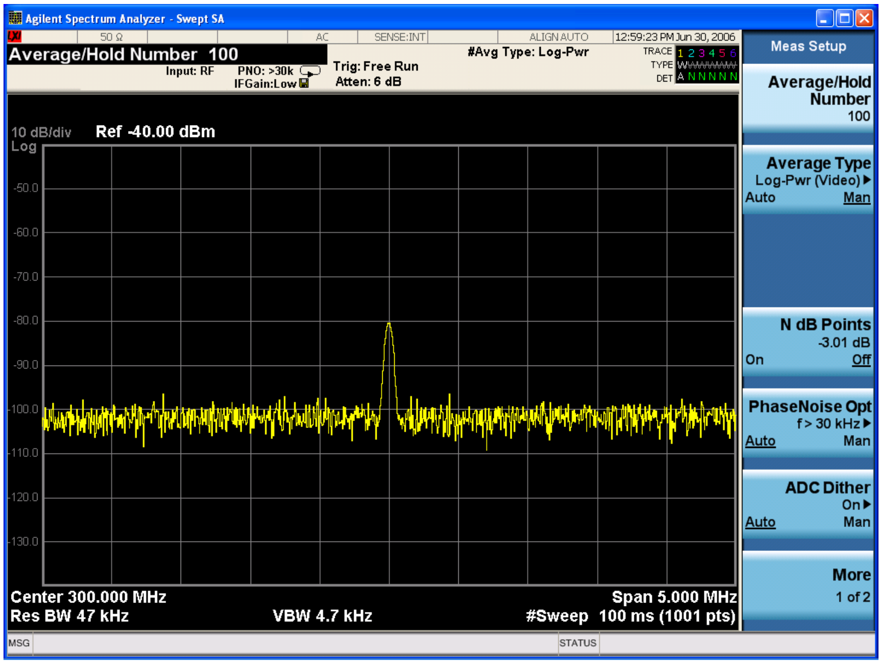
Note how the noise smooths out, as there is more time to average the values for each of the displayed data points.

Step 8. With the sweep time at 100 ms, change the average type to log averaging:

Press **Meas Setup**, **Average Type**, **Log-Pwr Avg** (Video).

Note how the noise level drops.

Figure 4-8 Varying the Sweep Time with the Average Detector



Trace Averaging

Averaging is a digital process in which each trace point is averaged with the previous average for the same trace point. Selecting averaging, when the analyzer is autocoupled, changes the detection mode from normal to sample. Sample mode may not measure a signal's amplitude as accurately as normal mode, because it may not find the true peak.

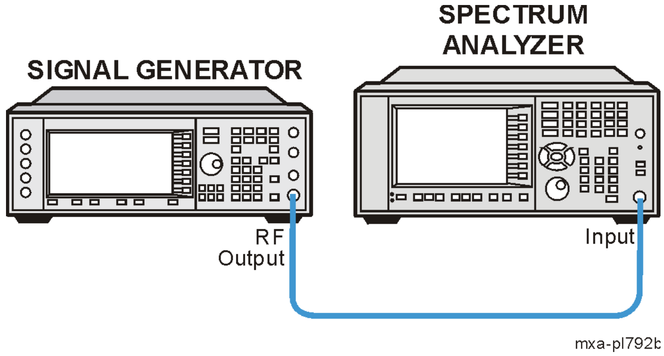
NOTE This is a trace processing function and is not the same as using the average detector (as described on page 54).

Step 1. Setup the signal sources as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –80 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 4-7.

Figure 4-9 **Setup for Measuring a Low-Level Signal**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the center frequency, span and reference level:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 5, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 40, **–dBm**.

Step 6. Turn trace averaging on:

Press **Trace/Detector**, **Trace Average**.

As the averaging routine smooths the trace, low level signals become more visible. `Average/Hold Number 100` appears in the measurement bar near the top of the screen. Refer to Figure 4-10

Step 7. With trace average as the active function, set the number of averages to 25:

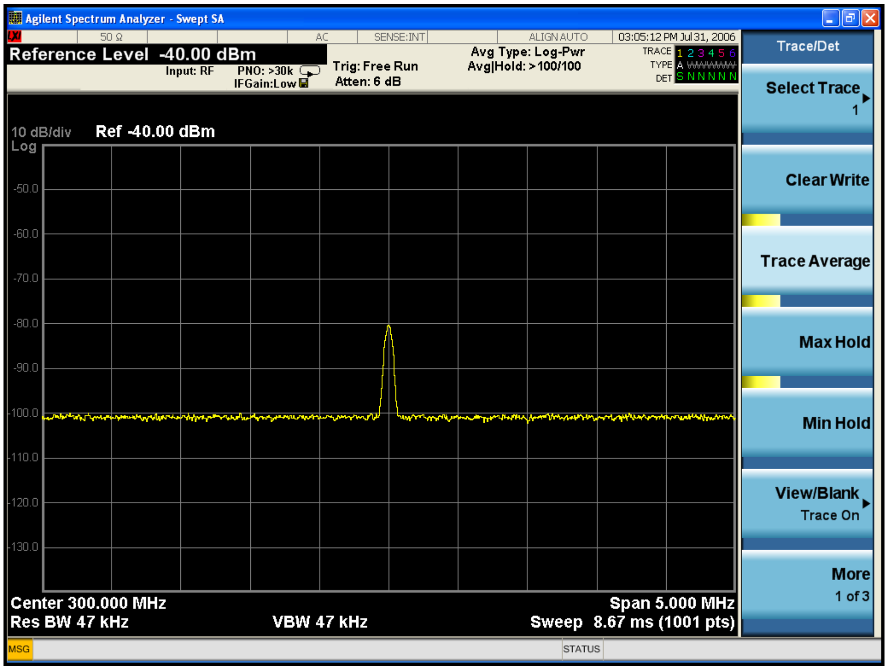
Press **Meas Setup**, **Average/Hold Number**, 25, **Enter**.

Annotation above the graticule in the measurement bar to the right of center shows the type of averaging, `Log-Power`. Also, the number of traces averaged is shown on the Average/Hold Number key.

Changing most active functions restarts the averaging, as does pressing the **Restart** key. Once the set number of sweeps completes, the analyzer continues to provide a running average based on this set number.

NOTE If you want the measurement to stop after the set number of sweeps, use single sweep: Press **Single** and then press the **Restart** key.

Figure 4-10 Trace Averaging



5 Improving Frequency Resolution and Accuracy

Using a Frequency Counter to Improve Frequency Resolution and Accuracy

This procedure uses the signal analyzer internal frequency counter to increase the resolution and accuracy of the frequency readout.

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Enable the internal 50 MHz amplitude reference signal as follows:

Press **Input/Output**, **RF Calibrator**, **50 MHz**.

Step 4. Set the center frequency to 50 MHz and the span to 80 MHz:

Press **FREQ Channel**, **Center Freq**, 50, **MHz**.
Press **SPAN X Scale**, **Span**, 80, **MHz**.

Step 5. Turn the frequency counter on:

Press **Marker**, **More**, **Marker Count**, **Marker Count** (On).

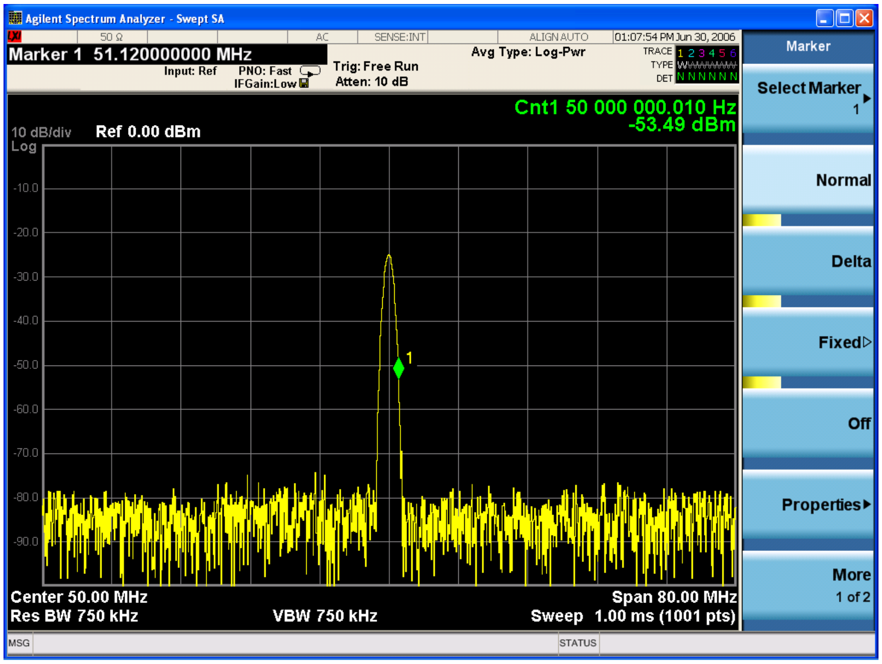
Step 6. Move the marker, with the front-panel knob, half-way down the skirt of the signal response.

Press **Marker**, rotate front-panel knob.

Notice that the readout in the active frequency function changes while the counted frequency result (upper-right corner of display) does not. See Figure 5-1. To get an accurate count, you do not need to place the marker at the exact peak of the signal response.

NOTE Marker count properly functions only on CW signals or discrete spectral components. The marker must be > 25 dB above the displayed noise level.

Figure 5-1 Using Marker Counter



NOTE If the **Gate Time** (under the **Marker Count** menu) is an integer multiple of the length of a power-line cycle (20 ms for 50 Hz power, 16.67 ms for 60 Hz power), the counter rejects incidental modulation at the power line rate. The shortest **Gate Time** that rejects both 50 and 60 Hz modulation is 100 ms (100 ms is the default **Gate Time** setting when set to **Auto**).

Step 7. The marker counter remains on until turned off. Turn off the marker counter:

Press **Marker**, **More**, **Marker Count**, **Marker Count** (Off)
Or
Press **Marker**, **Off**.

6 Tracking Drifting Signals

Measuring a Source's Frequency Drift

The analyzer can measure the short- and long-term stability of a source. The maximum amplitude level and the frequency drift of an input signal trace can be displayed and held by using the maximum-hold function. You can also use the maximum hold function if you want to determine how much of the frequency spectrum a signal occupies.

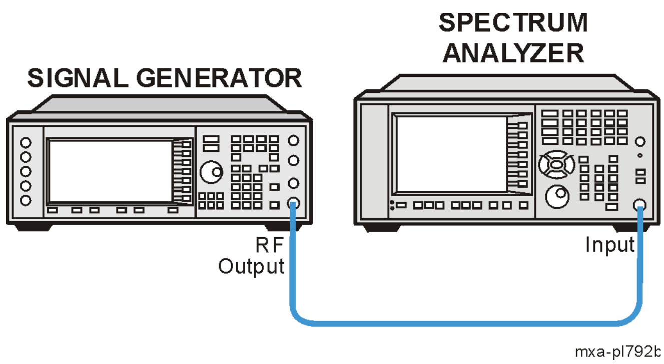
This procedure using signal tracking to keep the drifting signal in the center of the display. The drifting is captured by the analyzer using maximum hold.

Step 1. Setup the signal sources as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –20 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 6-1.

Figure 6-1 **Setup for Measuring Source Drift**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency, span and reference level.

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 10, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, 10, –**dBm**.

Step 6. Place a marker on the peak of the signal and turn signal tracking on:

Press **Peak Search**.
Press **SPAN X Scale**, **Signal Track** (On).

Step 7. Reduce the span to 500 kHz:

Press **SPAN**, 500, **kHz**.

Notice that the signal is held in the center of the display.

Step 8. Turn off the signal track function:

Press **SPAN X Scale**, **Signal Track** (Off).

Step 9. Measure the excursion of the signal with maximum hold:

Press **Trace/Detector**, **Max Hold**.

As the signal varies, maximum hold maintains the maximum responses of the input signal.

NOTE Annotation in the Trace/Detector panel, upper right corner of the screen, indicates the trace mode. In this example, the `M` in the Type row under TRACE `1` indicates trace 1 is in maximum-hold mode.

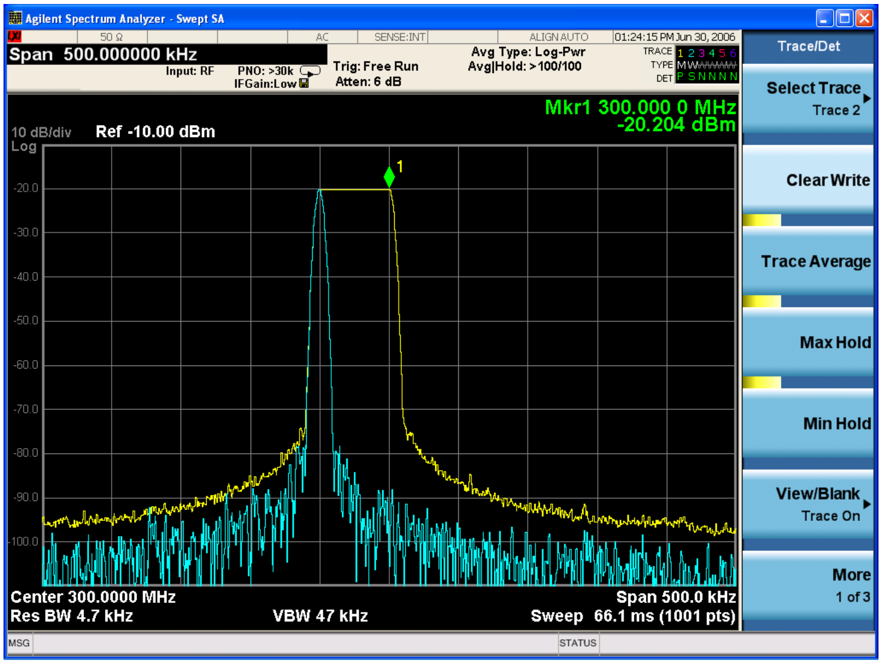
Step 10. Activate trace 2 and change the mode to continuous sweeping:

Press **Trace/Detector**, **Select Trace**, **Trace** (2).
Press **Clear Write**.

Trace 1 remains in maximum hold mode to show any drift in the signal.

Step 11. Slowly change the frequency of the signal generator ± 50 kHz in 1 kHz increments. Your analyzer display should look similar to Figure 6-2.

Figure 6-2 Viewing a Drifting Signal With Max Hold and Clear Write



Tracking a Signal

The signal track function is useful for tracking drifting signals that drift relatively slowly by keeping the signal centered on the display as the signal drifts. This procedure tracks a drifting signal.

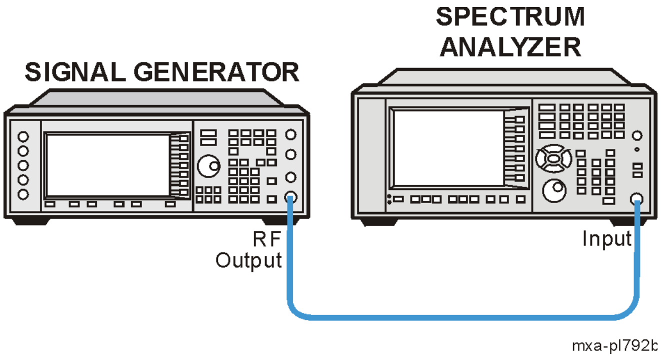
Note that the primary function of the signal track function is to track unstable signals, not to track a signal as the center frequency of the analyzer is changed. If you choose to use the signal track function when changing center frequency, check to ensure that the signal found by the tracking function is the correct signal.

Step 1. Setup the signal sources as follows:

Set the frequency of the signal source to 300 MHz.
Set the source amplitude to –20 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 6-3.

Figure 6-3 **Setup for Tracking a Signal**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency at a 1 MHz offset:

Press **FREQ Channel**, **Center Freq**, 301, **MHz**.
Press **SPAN X Scale**, **Span**, 10, **MHz**.

Step 6. Turn the signal tracking function on:

Press **SPAN X Scale**, **Signal Track** (On).

Notice that signal tracking places a marker on the highest amplitude peak and then brings the selected peak to the center of the display.

After each sweep the center frequency of the analyzer is adjusted to keep the selected peak in the center.

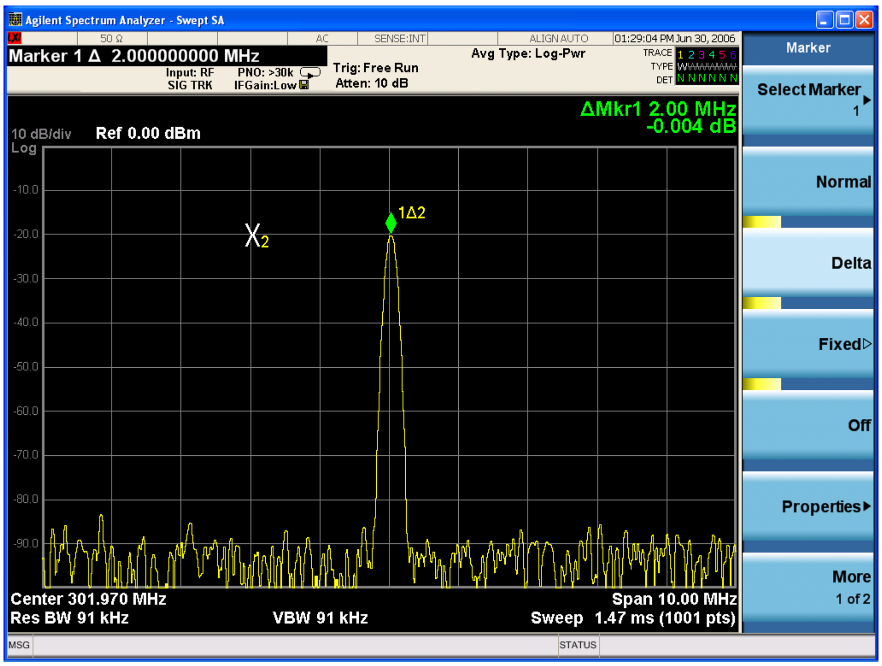
Step 7. Turn the delta marker on to read signal drift:

Press **Marker**, **Delta**.

Step 8. Tune the frequency of the signal generator in 100 kHz increments.

Notice that the center frequency of the analyzer also changes in 100 kHz increments, centering the signal with each increment.

Figure 6-4 **Tracking a Drifting Signal**



7 Making Distortion Measurements

Identifying Analyzer Generated Distortion

High level input signals may cause internal analyzer distortion products that could mask the real distortion measured on the input signal. Using trace 2 and the RF attenuator, you can determine which signals, if any, are internally generated distortion products.

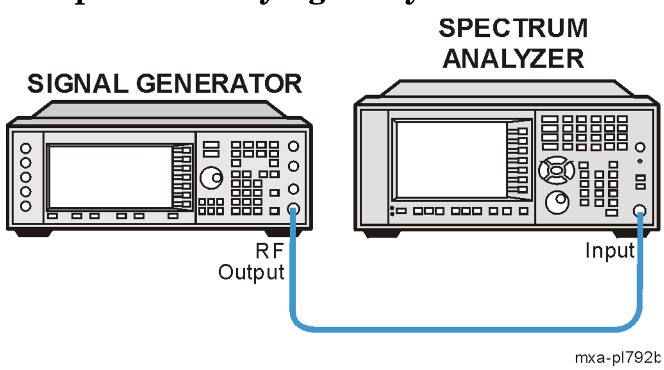
Using a signal from a signal generator, determine whether the harmonic distortion products are generated by the analyzer.

Step 1. Setup the signal source as follows:

Set the frequency of the signal source to 200 MHz.
Set the source amplitude to 0 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 7-1.

Figure 7-1 Setup for Identifying Analyzer Generated Distortion



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

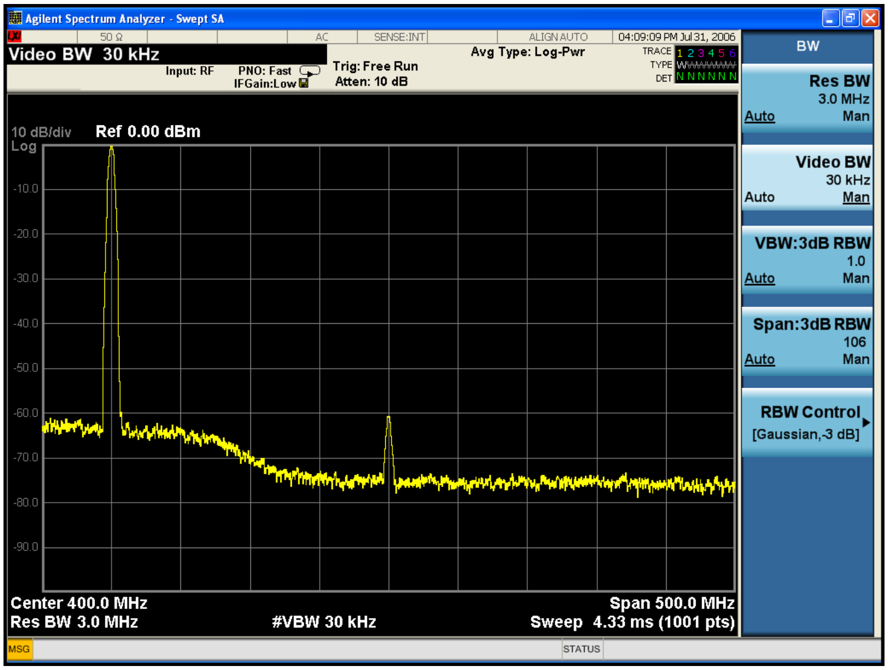
Press **Mode Preset**.

Step 5. Set the analyzer center frequency, span, and video bandwidth:

Press **FREQ Channel**, **Center Freq**, 400, **MHz**.
Press **SPAN X Scale**, **Span**, 500, **MHz**.
Press **BW**, **Video BW**, 30, **kHz**.

The signal produces harmonic distortion products (spaced 200 MHz from the original 200 MHz signal) in the analyzer input mixer as shown in Figure 7-2.

Figure 7-2 **Harmonic Distortion**



Step 6. Change the center frequency to the value of the first harmonic:

Press **Peak Search**, **Next Peak**, **Mkr→CF**.

Step 7. Change the span to 50 MHz and re-center the signal:

Press **SPAN X Scale**, **Span**, 50, **MHz**.
Press **Peak Search**, **Mkr→CF**.

Step 8. Set the attenuation to 0 dB:

Press **AMPTD Y Scale**, **Attenuation**, 0, **dB**.

Step 9. To determine whether the harmonic distortion products are generated by the analyzer, first save the trace data in trace 2 as follows:

Press **Trace/Detector**, **Select Trace**, **Trace 2**, **Clear Write**.

Step 10. Allow trace 2 to update (minimum two sweeps), then store the data from trace 2 and place a delta marker on the harmonic of trace 2:

Press **Trace/Detector**, **View/Blank**, **View** (Update Off, Display On).
Press **Peak Search**, **Marker Delta**.

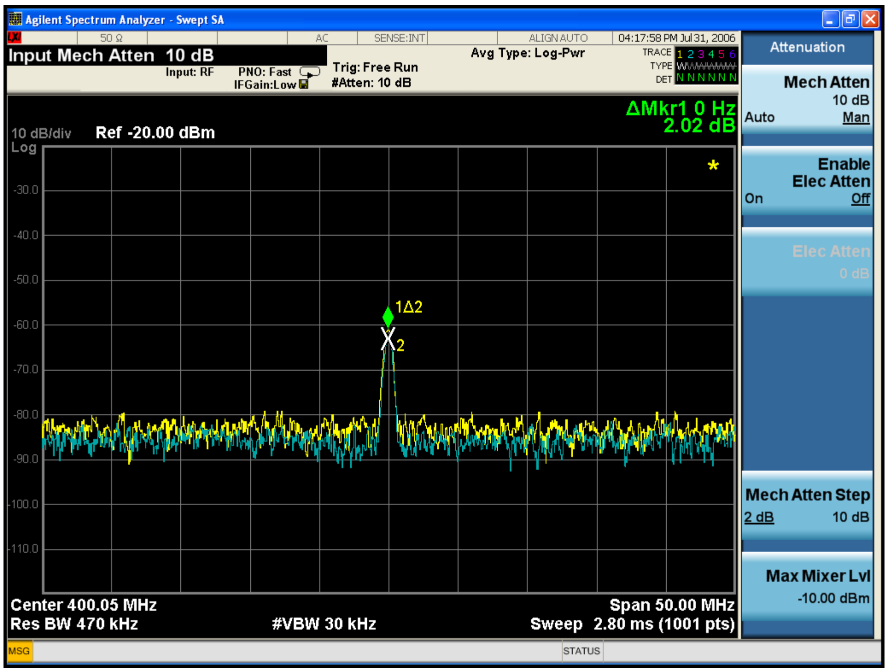
The analyzer display shows the stored data in trace 2 and the measured data in trace 1. The ΔMkr1 amplitude reading is the difference in amplitude between the reference and active markers.

Step 11. Increase the RF attenuation to 10 dB:

Press **AMPTD Y Scale**, **Attenuation**, **10**, **dB**.

Notice the ΔMkr1 amplitude reading. This is the difference in the distortion product amplitude readings between 0 dB and 10 dB input attenuation settings. If the ΔMkr1 amplitude absolute value is approximately ≥1 dB for an input attenuator change, the distortion is being generated, at least in part, by the analyzer. In this case more input attenuation is necessary. Increase the input attenuation until ΔMkr1 amplitude stops increasing or decreasing in value. Return to the previous attenuator step and the input signal distortion measured will be minimally impacted by the analyzer internally generated distortion. See Figure 7-3.

Figure 7-3 RF Attenuation of 10 dB



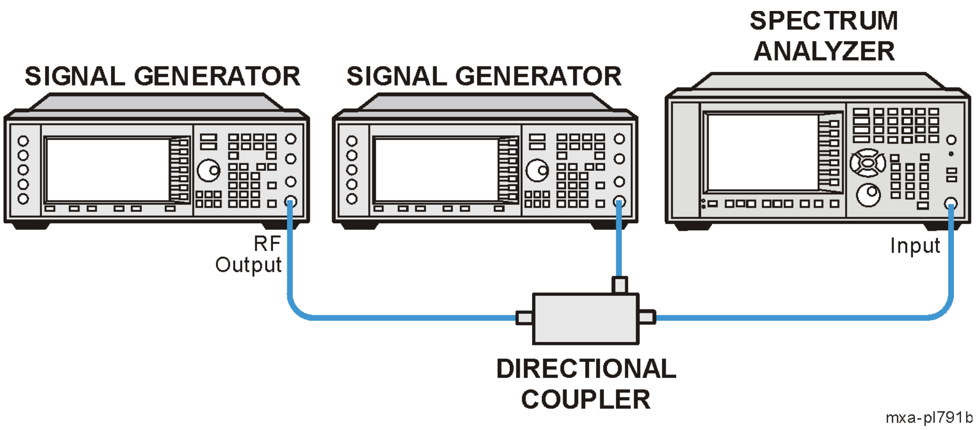
Third-Order Intermodulation Distortion

Two-tone, third-order intermodulation distortion is a common test in communication systems. When two signals are present in a non-linear system, they can interact and create third-order intermodulation distortion products that are located close to the original signals. These distortion products are generated by system components such as amplifiers and mixers.

This procedure tests a device for third-order intermodulation using markers. Two sources are used, one set to 300 MHz and the other to 301 MHz.

Step 1. Connect the equipment as shown in Figure 7-4. This combination of signal generators, low pass filters, and directional coupler (used as a combiner) results in a two-tone source with very low intermodulation distortion. Although the distortion from this setup may be better than the specified performance of the analyzer, it is useful for determining the TOI performance of the source/analyzer combination. After the performance of the source/analyzer combination has been verified, the device-under-test (DUT) (for example, an amplifier) would be inserted between the directional coupler output and the analyzer input.

Figure 7-4 Third-Order Intermodulation Equipment Setup



NOTE The coupler should have a high degree of isolation between the two input ports so the sources do not intermodulate.

Step 2. Set the sources as follows:

Set one signal generator to 300 MHz.
Set the other source to 301 MHz. This produces a frequency separation of 1 MHz.
Set the sources equal in amplitude as measured by the analyzer (in this example, they are set to –5 dBm).

Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency and span:

Press **FREQ Channel**, **Center Freq**, 300.5, **MHz**.
Press **SPAN X Scale**, **Span**, 5, **MHz**.

Step 6. Set the analyzer detector to Peak:

Press **Trace/Detector**, **Detector**, **Peak**.

Step 7. Set the mixer level to improve dynamic range:

Press **AMPTD Y Scale**, **Attenuation**, **Max Mixer Lvl**, –10, **dBm**.

The analyzer automatically sets the attenuation so that a signal at the reference level has a maximum value of –10 dBm at the input mixer.

Step 8. Move the signal to the reference level:

Press **Peak Search**, **Mkr** →, **Mkr** → **Ref Lvl**.

Step 9. Reduce the RBW until the distortion products are visible:

Press **BW**, **Res BW**, ↓.

Step 10. Activate the second marker and place it on the peak of the distortion product (beside the test signal) using the **Next Right** or **Next Left** key.

Press **Peak Search**, **Marker Delta**, **Next Left**.

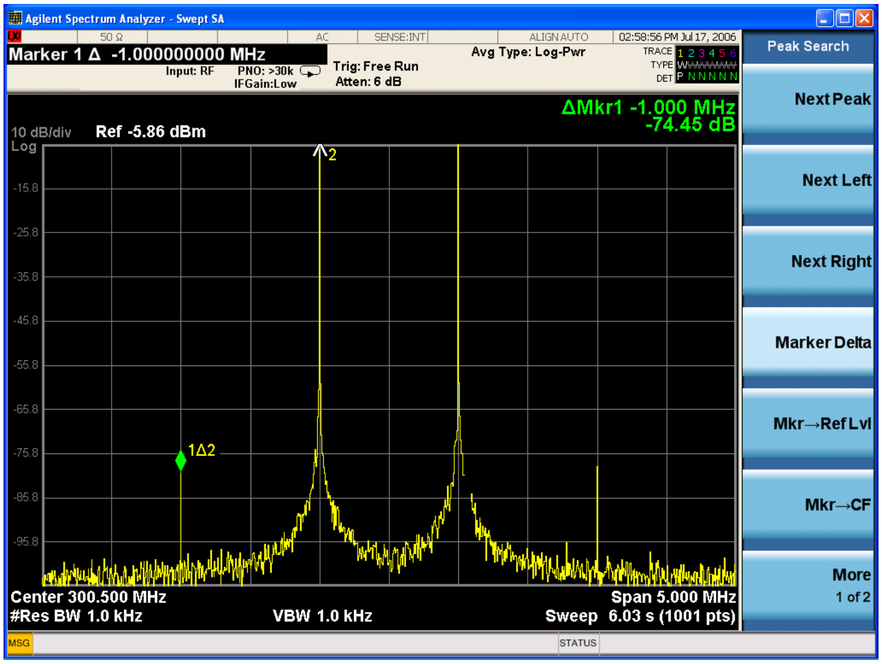
Step 11. Measure the other distortion product:

Press **Marker**, **Normal**, **Peak Search**, **Next Peak**.

Step 12. Measure the difference between this test signal and the second distortion product (see Figure 7-5):

Press **Peak Search**, **Marker Delta**, **Next Right**.

Figure 7-5 Measuring the Distortion Product



8 Measuring Noise

Measuring Signal-to-Noise

Signal-to-noise is a ratio used in many communication systems as an indication of noise in a system. Typically the more signals added to a system adds to the noise level, reducing the signal-to-noise ratio making it more difficult for modulated signals to be demodulated. This measurement is also referred to as carrier-to-noise in some communication systems.

The signal-to-noise measurement procedure below may be adapted to measure any signal in a system if the signal (carrier) is a discrete tone. If the signal in your system is modulated, it is necessary to modify the procedure to correctly measure the modulated signal level.

In this example the 50 MHz amplitude reference signal is used as the fundamental signal. The amplitude reference signal is assumed to be the signal of interest and the internal noise of the analyzer is measured as the system noise. To do this, you need to set the input attenuator such that both the signal and the noise are well within the calibrated region of the display.

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Enable the internal 50 MHz amplitude reference signal as follows:

Press **Input/Output**, **RF Calibrator**, 50, **MHz**.

Step 4. Set the center frequency, span, reference level and attenuation:

Press **FREQ Channel**, **Center Freq**, 50, **MHz**.
Press **SPAN X Scale**, **Span**, 1, **MHz**.
Press **AMPTD Y Scale**, **Ref Level**, –10, **dBm**.
Press **AMPTD Y Scale**, **Attenuation**, 40, **dB**.

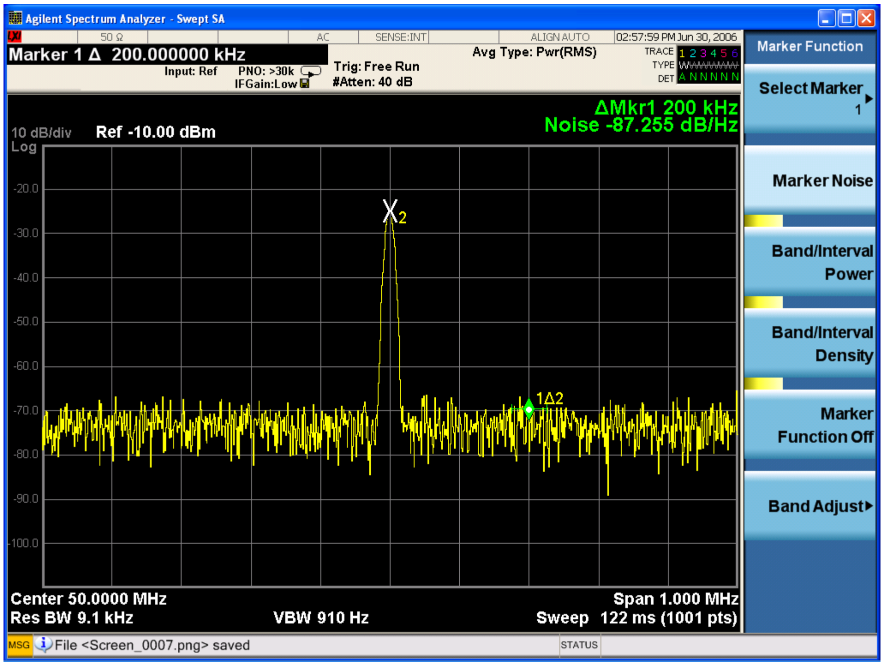
Step 5. Place a marker on the peak of the signal and then place a delta marker in the noise at a 200 kHz offset:

Press **Peak Search**, **Marker Delta**, 200, **kHz**.

Step 6. Turn on the marker noise function to view the signal-to-noise measurement results:

Press **Marker Function**, **Marker Noise**.

Figure 8-1 **Measuring the Signal-to-Noise**



Read the signal-to-noise in dB/Hz, that is with the noise value determined for a 1 Hz noise bandwidth. If you wish the noise value for a different bandwidth, decrease the ratio by $10 \times \log(BW)$. For example, if the analyzer reading is –70 dB/Hz but you have a channel bandwidth of 30 kHz:

$$\text{S/N} = -70\ \text{dB/Hz} + 10 \times \log(30\ \text{kHz}) = -25.23\ \text{dB}/(30\ \text{kHz})$$

NOTE The display detection mode is now average. If the delta marker is closer than one quarter of a division away from the edge of the response to the discrete signal, the amplitude reference signal in this case, there is a potential for error in the noise measurement. See “Measuring Noise Using the Noise Marker” on page 80.

Measuring Noise Using the Noise Marker

This procedure uses the marker function, **Marker Noise**, to measure noise in a 1 Hz bandwidth. In this example the noise marker measurement is made near the 50 MHz reference signal to illustrate the use of **Marker Noise.**

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Enable the internal 50 MHz amplitude reference signal as follows:

Press **Input/Output**, **RF Calibrator**, 50, **MHz**.
Press **FREQ Channel**, **Center Freq**, 49.98, **MHz**.
Press **Span X Scale**, **Span**, 100, **kHz**.
Press **AMPTD Y Scale**, **Ref Level**, –10, **dBm**.
Press **AMPTD Y Scale**, **Attenuation**, **Mech Atten**, 40, **dBm**.

Step 4. Activate the noise marker:

Press **Marker Function**, **Marker Noise**.

Note that display detection automatically changes to “Avg”; average detection calculates the noise marker from an average value of the displayed noise. Notice that the noise marker floats between the maximum and the minimum displayed noise points. The marker readout is in dBm (1 Hz) or dBm per unit bandwidth.

For noise power in a different bandwidth, add $10 \times \log(\mathrm{BW})$. For example, for noise power in a 1 kHz bandwidth, dBm (1 kHz), add $10 \times \log(1000)$ or 30 dB to the noise marker value.

Step 5. Reduce the variations of the sweep-to-sweep marker value by increasing the sweep time:

Press **Sweep/Control**, **Sweep Time**, 3, **s**.

Increasing the sweep time when the average detector is enabled allows the trace to average over a longer time interval, thus reducing the variations in the results (increases measurement repeatability).

Step 6. Move the marker to 50 MHz:

Press **Marker**, 50, **MHz**.

The noise marker value is based on the mean of 5% of the total number of sweep points centered at the marker in the initially selected span.

The points that are averaged span one-half of a division. Changing spans after enabling the noise marker will result in the marker averaging a progressively wider or narrower portion of the newly selected span and corresponding sweep points. This occurs because the marker is locked to 5% of the initially selected span.

Step 7. To adjust the width of the noise marker relative to the span:

Press **Marker Function**, **Band Adjust**, **Band/Interval Span**, and adjust the value to the desired marker width.

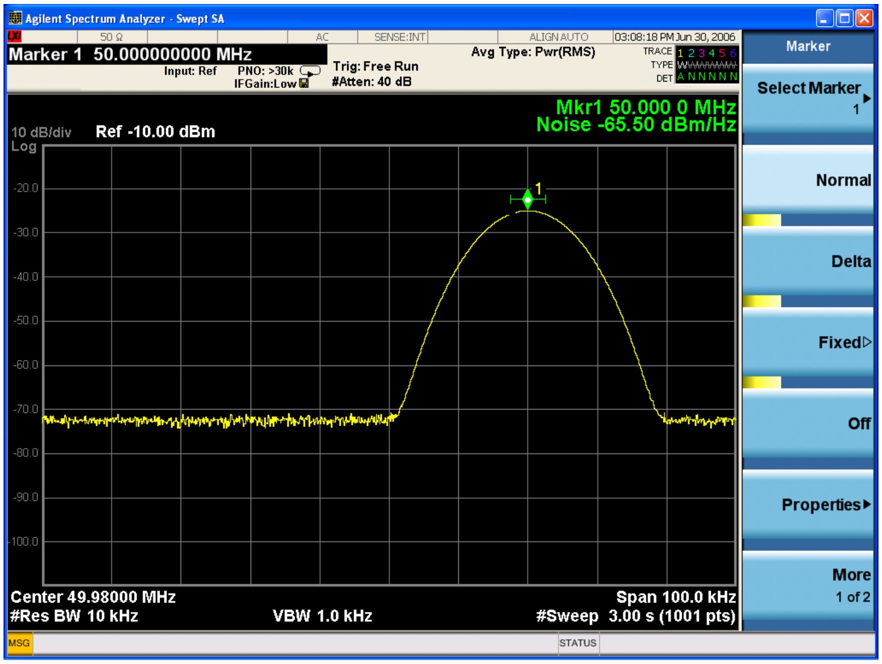
Notice that the marker does not go to the peak of the signal because there are not enough points at the peak of the signal. The noise marker is also averaging points below the peak due to the narrow RBW.

Step 8. Widen the resolution bandwidth to allow the marker to make a more accurate peak power measurement using the noise marker:

Press **BW**, **Res BW**, 10, **kHz**.

Refer to Figure 8-2.

Figure 8-2 Noise Marker



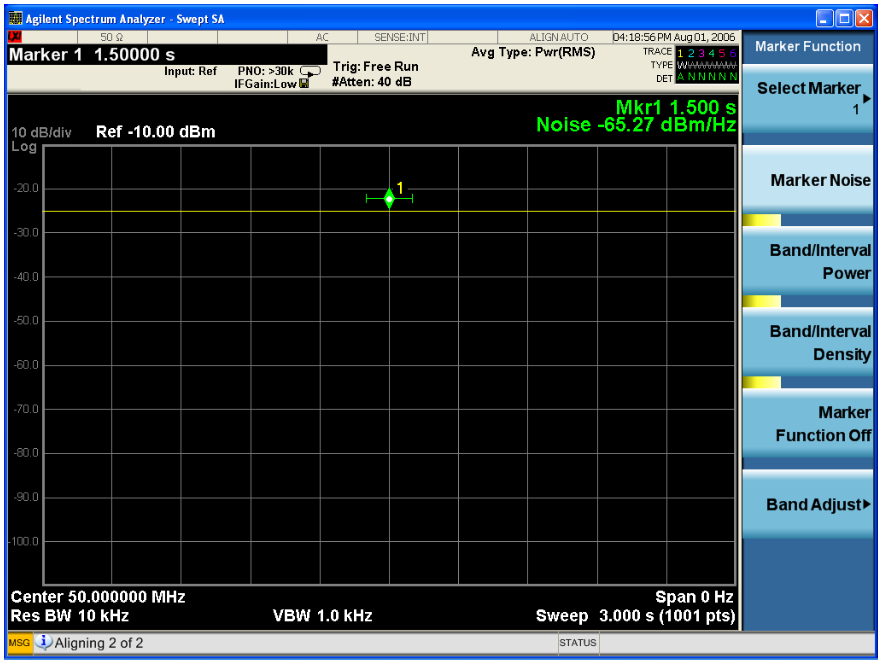
Step 9. Set the analyzer to zero span at the marker frequency:

Press **Mkr** →, **Mkr** → **CF**.
Press **SPAN X Scale**, **Zero Span**.
Press **Marker**, **Function**, **Marker Noise**.

Note that the marker amplitude value is now correct since all points averaged are at the same frequency and not influenced by the shape of the bandwidth filter. (See Figure 8-3.)

Remember that the noise marker calculates a value based on an average of the points around the frequency of interest. Generally when making power measurements using the noise marker on discrete signals, first tune to the frequency of interest and then make your measurement in zero span (time-domain).

Figure 8-3 Noise Marker with Zero Span



Measuring Noise-Like Signals Using Band/Interval Density Markers

Band/Interval Density Markers let you measure power over a frequency span. The markers allow you to easily and conveniently select any arbitrary portion of the displayed signal. However, while the analyzer, when autocoupled, makes sure the analysis is power-responding (rms voltage-responding), you must set all of the other parameters.

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Set the center frequency, span, reference level and attenuation:

Press **FREQ Channel**, **Center Freq**, 50, **MHz**.
Press **SPAN X Scale**, **Span**, 100, **kHz**.
Press **AMPTD Y Scale**, **Ref Level**, –20, **dBm**.
Press **AMPTD Y Scale**, **Attenuation**, 40, **dB**.

Step 4. Measure the total noise power between the markers:

Press **Marker Function**, **Band/Interval Density**.

Step 5. Set the band span:

Press **Band Adjust**, **Band/Interval Span**, 40, **kHz**.

Step 6. Set the resolution and video bandwidths:

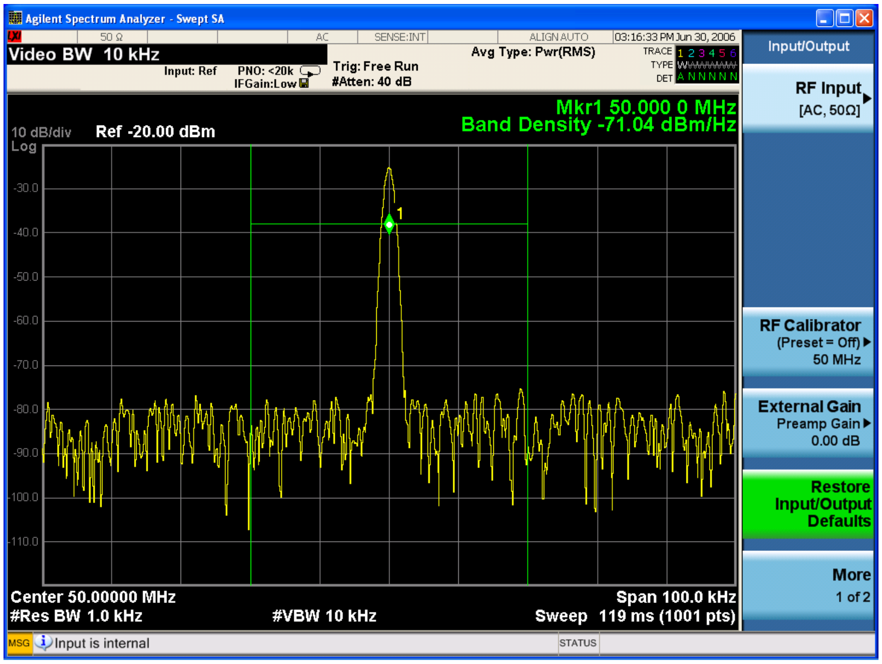
Press **BW**, **Res BW**, 1, **kHz**.
Press **BW**, **Video BW**, 10, **kHz**.

Common practice is to set the resolution bandwidth from 1% to 3% of the measurement (marker) span, 40 kHz in this example.

Step 7. Add a discrete tone to see the effects on the reading. Enable the internal 50 MHz amplitude reference signal of the analyzer as follows:

Press **Input/Output**, **RF Calibrator**, **50 MHz**.

Figure 8-4 Band/Interval Density Measurement

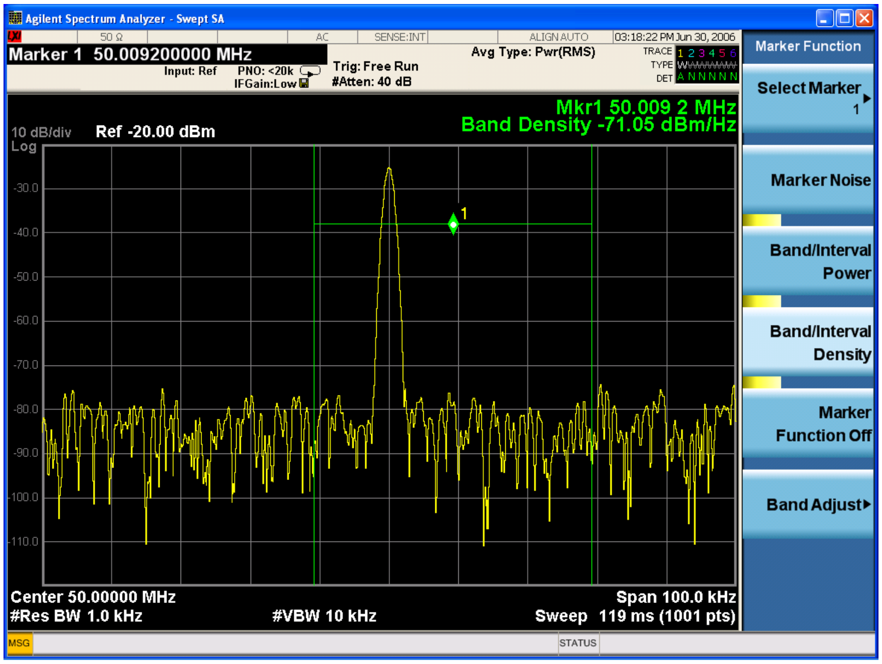


Step 8. Set the Band/Interval Density Markers to enable moving the markers (set at 40 kHz span) around without changing the Band/Interval span. Use the front-panel knob to move the band power markers and note the change in the power reading:

Press **Marker Function**, **Band/Interval Density**, then rotate front-panel knob. (Refer to Figure 8-5.)

NOTE Band/Interval Density Markers can be changed to read the total absolute power by pressing **Marker Function**, **Band/Interval Power**.

Figure 8-5 Band/Interval Density Measurement



Measuring Noise-Like Signals Using the Channel Power Measurement

You may want to measure the total power of a noise-like signal that occupies some bandwidth. Typically, channel power measurements are used to measure the total (channel) power in a selected bandwidth for a modulated (noise-like) signal. Alternatively, to manually calculate the channel power for a modulated signal, use the noise marker value and add $10 \times \log(\text{channel BW})$. However, if you are not certain of the characteristics of the signal, or if there are discrete spectral components in the band of interest, you can use the channel power measurement. This example uses the noise of the analyzer, adds a discrete tone, and assumes a channel bandwidth (integration bandwidth) of 2 MHz. If desired, a specific signal may be substituted.

Step 1. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 2. Preset the analyzer:

Press **Mode Preset**.

Step 3. Set the center frequency:

Press **FREQ Channel**, **Center Freq**, 50, **MHz**.

Step 4. Start the channel power measurement:

Press **Meas**, **Channel Power**.

Step 5. Enable the Bar Graph:

Press **View/Display**, **Bar Graph**, **On**.

Step 6. Add a discrete tone to see the effects on the reading. Enable the internal 50 MHz amplitude reference signal of the analyzer as follows:

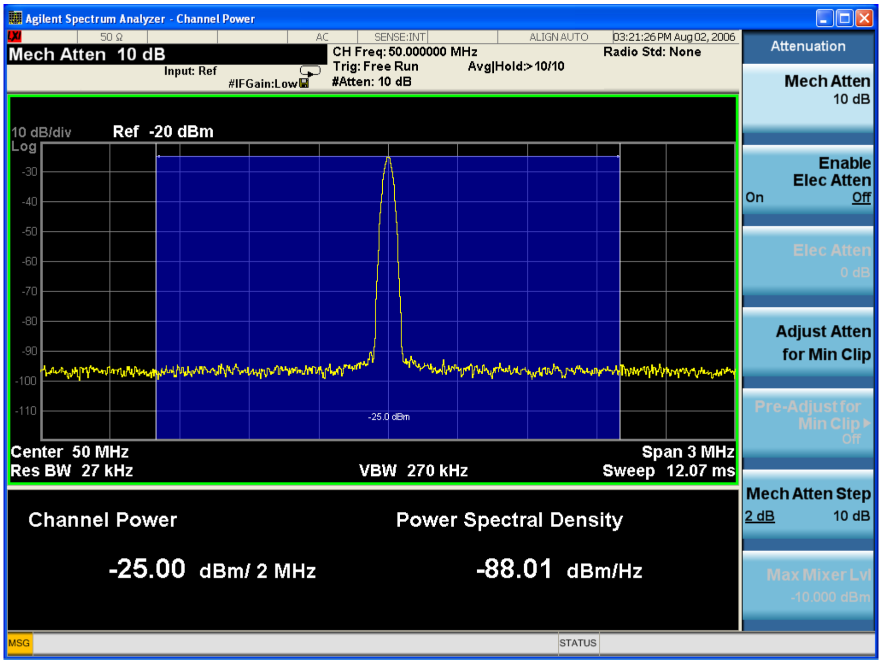
Press **Input/Output**, **RF Calibrator**, **50 MHz**.

Step 7. Optimize the analyzer attenuation level setting:

Press **AMPTD**, **Attenuation**, **Adjust Atten for Min Clip**.

Your display should be similar to Figure 8-6.

Figure 8-6 Measuring Channel Power



The power reading is essentially that of the tone; that is, the total noise power is far enough below that of the tone that the noise power contributes very little to the total.

The algorithm that computes the total power works equally well for signals of any statistical variant, whether tone-like, noise-like, or combination.

Measuring Signal-to-Noise of a Modulated Carrier

Signal-to-noise (or carrier-to-noise) is a ratio used in many communication systems as indication of the noise performance in the system. Typically, the more signals added to the system or an increase in the complexity of the modulation scheme can add to the noise level. This can reduce the signal-to-noise ratio and impact the quality of the demodulated signal. For example, a reduced signal-to-noise in digital systems may cause an increase in EVM (error vector magnitude).

With modern complex digital modulation schemes, measuring the modulated carrier requires capturing all of its power accurately. This procedure uses the Band Power Marker with a RMS average detector to correctly measure the carrier's power within a user adjustable region. A Noise Marker (normalized to a 1 Hz noise power bandwidth) with an adjustable noise region is also employed to allow the user to select and accurately measure just the system noise of interest. An important key to making accurate Band Power Marker and Noise Power measurements is to insure that the Average Type under the Meas Setup key is set to “Auto”.

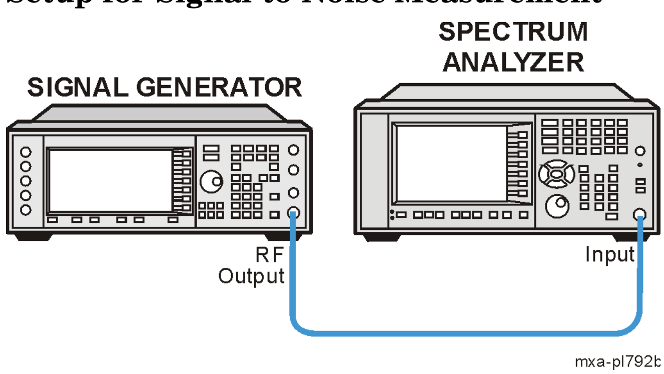
In this example a 4 carrier W-CDMA digitally modulated carrier is used as the fundamental signal and the internal noise of the analyzer is measured as the system noise.

Step 1. Setup the signal source as follows:

Setup a 4 carrier W-CDMA signal.
Set the source frequency to 1.96 GHz.
Set the source amplitude to –10 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 8-7.

Figure 8-7 Setup for Signal-to-Noise Measurement



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Tune to the W-CDMA signal:

Press **FREQ Channel**, **Auto Tune**

Step 6. Enable the Band Power Marker function for measuring the total power of the 4 carrier W-CDMA signal.

Press **Marker Function**, **Band/Interval Power**

Step 7. Center the frequency of the Band Power marker on the signal:

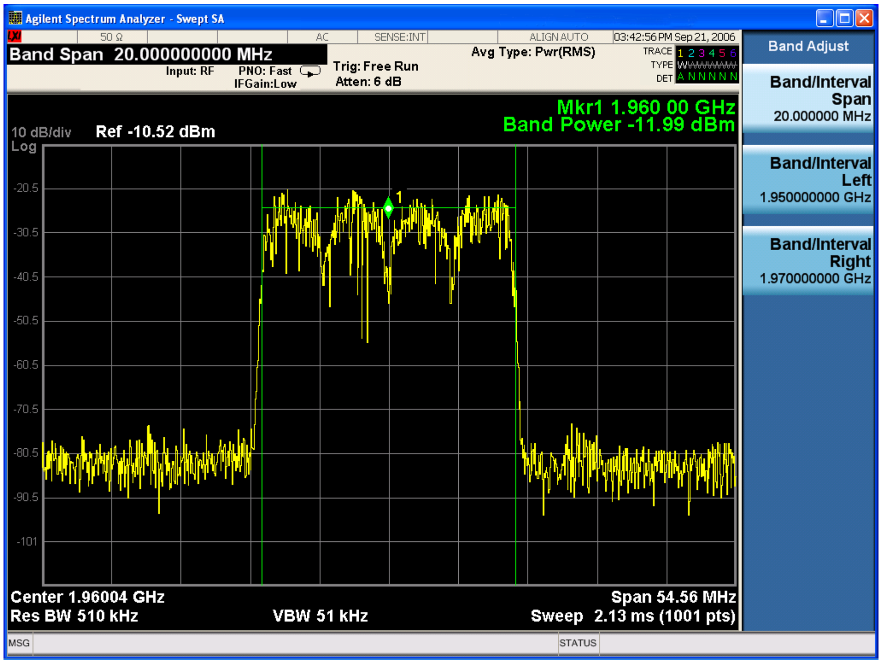
Press **Select Marker 1**, 1.96, **GHz**

Step 8. Adjust the width (or span) of the Band Power marker to encompass the entire 4 carrier W-CDMA signal (refer to Figure 8-8):

Note the green vertical lines of Marker 1 representing the span of signals included in the Band Power measurement and the carrier power indicated in Markers Result Block.

Press **Marker Function**, **Band Adjust**, **Band/Interval Span**, 20, **MHz**

Figure 8-8 **4 Carrier W-CDMA Signal Power using Band Power Marker**



Step 9. Enable the Noise Marker using marker 2 for measuring the system noise power:

Press **Marker Function**, **Select Marker**, **Marker 2**, **Marker Noise**

Step 10. Move the Noise Marker 2 to the system noise frequency of interest:

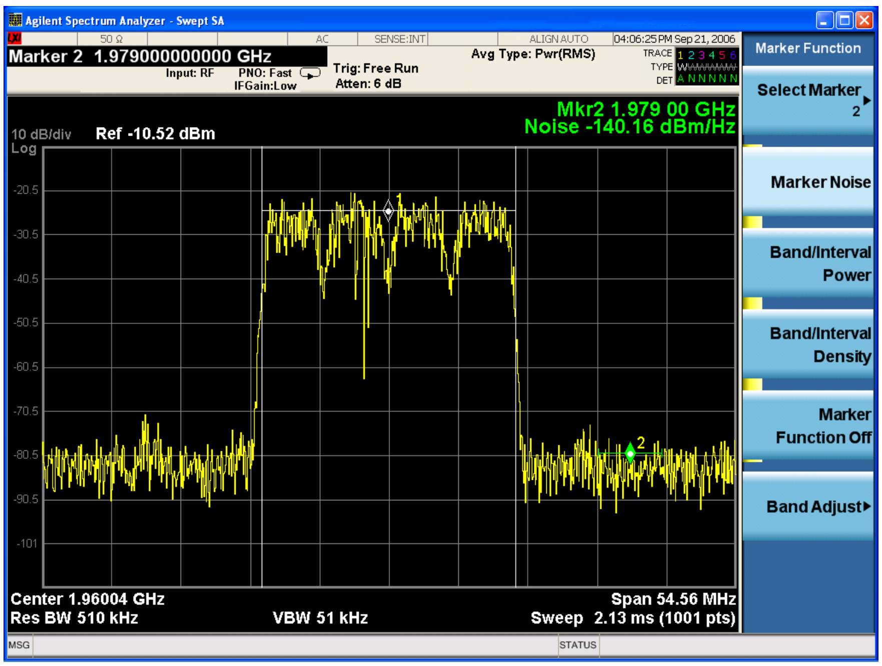
Press **Select Marker 2**, 1.979, **GHz**

Step 11. Adjust the width of the noise marker region to encompass the desired noise power (refer to Figure 8-9):

Note the green “wings” of Marker 2 outlining the noise region to be included in the measurement and the resulting noise power expressed in dBm/Hz as shown in the Marker Results Block.

Press **Marker Function**, **Select Marker**, **Marker 2**, **Band Adjust**, **Band/Interval**, 5, **MHz**

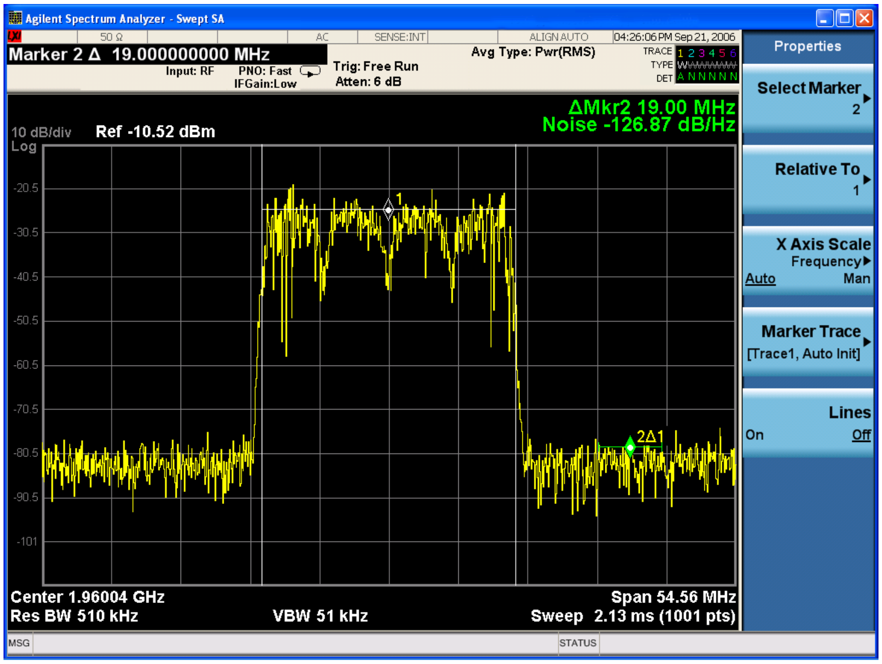
Figure 8-9 Noise Marker Measuring System Noise



Step 12. Measure carrier-to-noise by making the Noise Marker relative to the carrier's Band Power Marker (refer to Figure 8-10):

Press **Marker**, **Properties**, **Select Marker**, **Marker 2**, **Relative to**, **Marker 1**

Figure 8-10 Signal-to-Noise Measurement



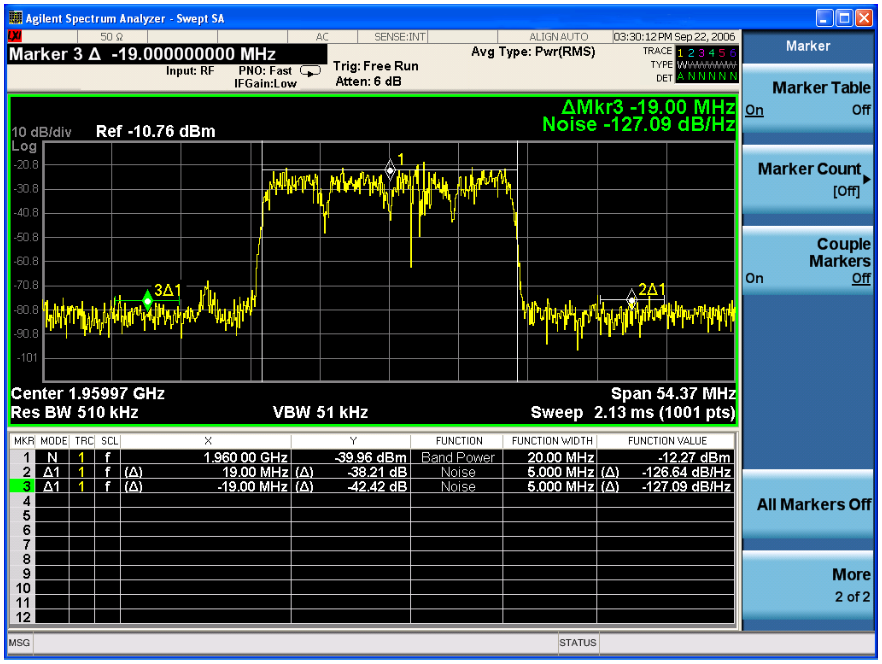
Step 13. Simultaneously measure carrier-to-noise on a second region of the system by enabling another Noise Marker (up to 11 available).

Press **Marker Function**, **Select Marker**, **Marker 3**, **Marker Noise**
Press **Select Marker 3**, 1.941, **GHz**
Press **Return**, **Band Adjust**, **Band/Interval**, 5, **MHz**
Press **Marker**, **Properties**, **Select Marker**, **Marker 3**, **Relative to**, **Marker 1**

Step 14. Enable the Marker Table to view results of both carrier-to-noise measurements and all other markers (refer to Figure 8-10):

Press **Marker**, **More**, **Marker Table**, **On**

Figure 8-11 **Multiple Signal-to-Noise Measurements with Marker Table**



MKR	MODE	TRC	SCL	X	Y	FUNCTION	FUNCTION WIDTH	FUNCTION VALUE
1	N	1	f	1.960 00 GHz	-39.96 dBm	Band Power	20.00 MHz	-12.27 dBm
2	Δ1	1	f (Δ)	19.00 MHz (Δ)	-38.21 dB	Noise	5.000 MHz (Δ)	-126.64 dB/Hz
3	Δ1	1	f (Δ)	-19.00 MHz (Δ)	-42.42 dB	Noise	5.000 MHz (Δ)	-127.09 dB/Hz

Improving Phase Noise Measurements by Subtracting Signal Analyzer Noise

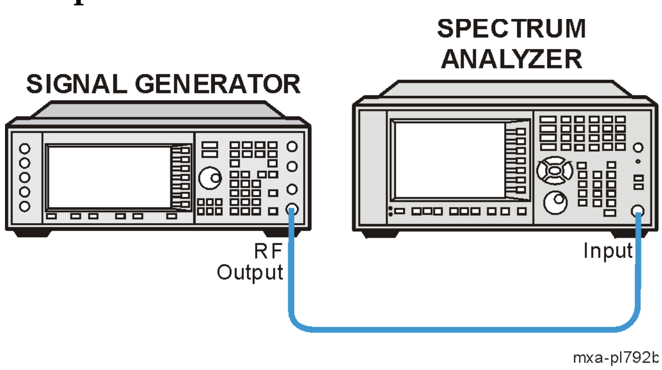
Making noise power measurements (such as phase noise) near the noise floor of the signal analyzer can be challenging where every db improvement is important. Utilizing the MXA trace math function Power Diff and 3 separate traces allows measurement of the DUT's phase noise in one trace, the analyzers noise floor in a second trace and then the resulting subtraction of those two traces displayed in a third trace with the analyzer's noise contribution removed.

Step 1. Setup the signal source as follows:

Setup an unmodulated signal
Set the source frequency to 1.96 GHz.
Set the source amplitude to –30 dBm

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 8-12.

Figure 8-12 Setup for Phase Noise Measurement



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

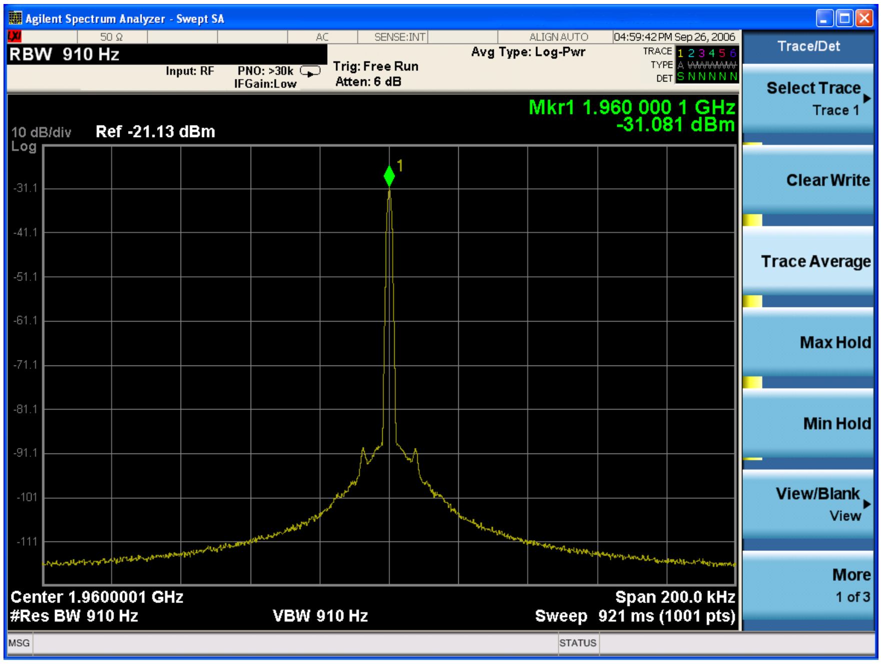
Step 5. Tune to the unmodulated carrier, adjust the span and RBW:

Press **FREQ Channel**, **Auto Tune**
Press **Span**, **Span**, 200, **kHz**
Press **BW**, **Res BW**, 910, **Hz**

Step 6. Measure and store the DUT phase noise plus MXA analyzer noise using trace 1 with trace averaging (allow time for sufficient averaging). Refer to Figure 8-13:

Press **Trace/Detector**, **Select Trace**, **Trace 1**, **Trace Average**
After sufficient averaging
Press **View/Blank**, **View**

Figure 8-13 Measurement of DUT and Analyzer Noise

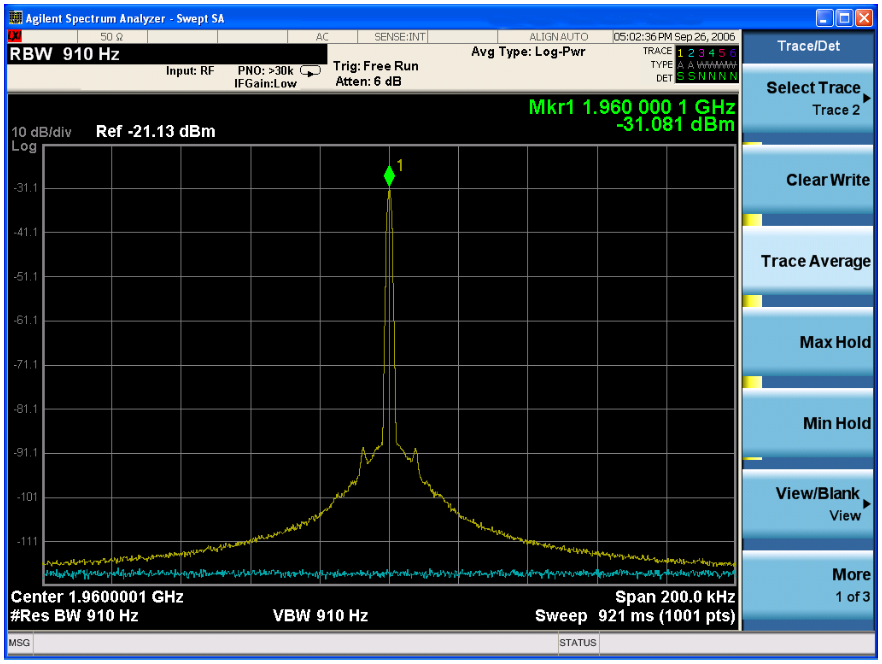


Step 7. Measure only the MXA analyzer noise using trace 2 (blue trace) with trace averaging (allow time for sufficient averaging). Refer to Figure 8-14:

Turn off or remove the DUT's signal to the RF input of the MXA analyzer

Press **Trace/Detector**, **Select Trace**, **Trace 2**, **Clear Write**, **Trace Average**
After sufficient averaging
Press **View/Blank**, **View**

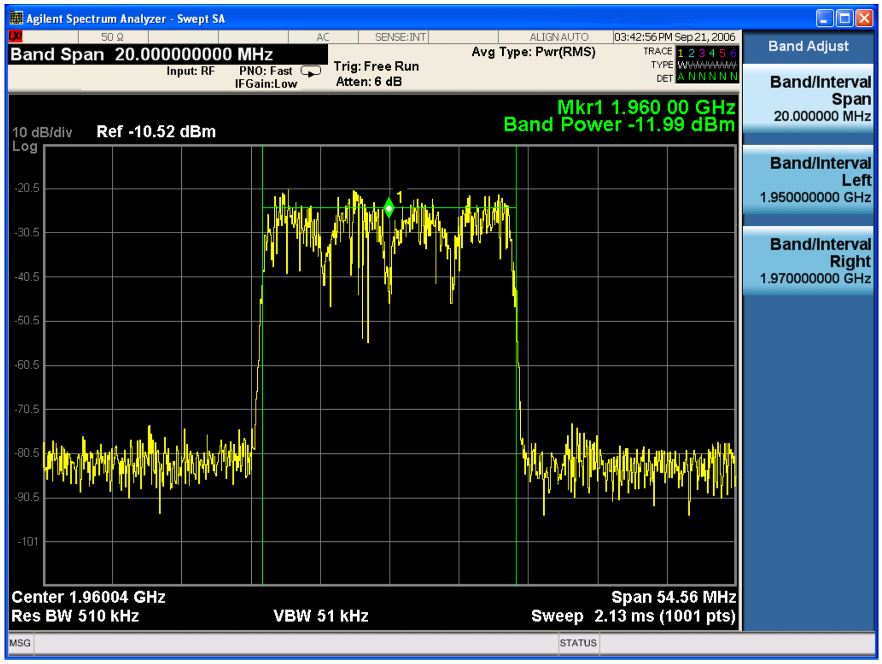
Figure 8-14 Measurement of Analyzer Noise



Step 8. Subtract the MXA noise from the DUT's phase noise measurement using the Power Diff math function and note the phase noise improvement at 100 kHz offset between trace 1 (yellow trace) and trace 3 (magenta trace). Refer to Figure 8-15:

Press **Trace/Detector**, **Select Trace**, **Trace 3**, **Clear Write**
Press **More**, **More**, **Math**, **Power Diff**, **Trace Operands**, **Operand 1**, **Trace 1**, **Operand 2**, **Trace 2**

Figure 8-15 Improved Phase Noise Measurement



Step 9. Measure the noise measurement improvement with delta Noise markers between traces. Note the up to 6 dB improvement in the Marker Results Block. Refer to Figure 8-16:

Press **Marker**, **Select Marker**, **Marker 1**, **Normal**
Press **Properties**, **Select Marker**, **Marker 1**, **Marker Trace**, **Trace 1**

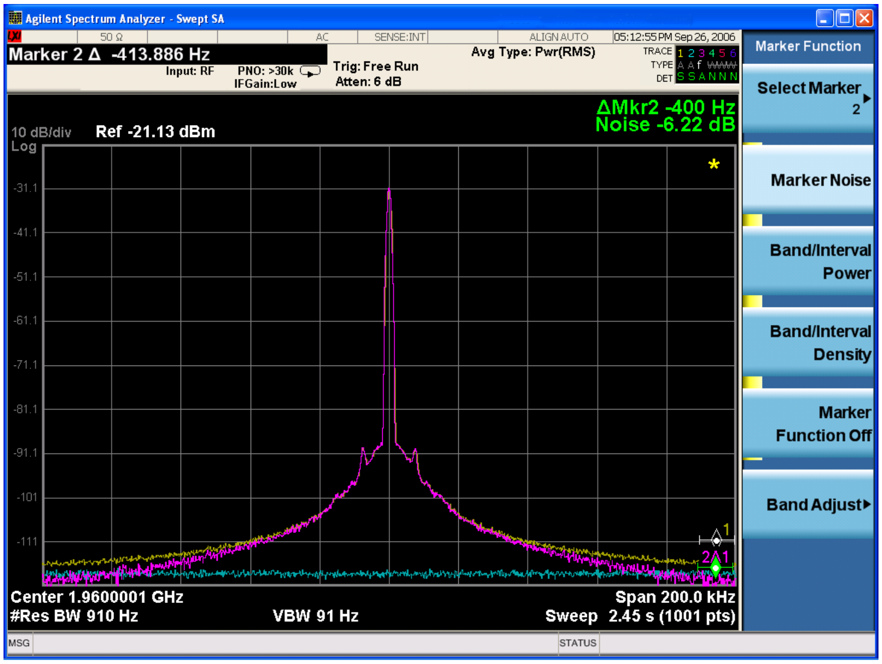
Adjust Marker 1 to approximately 200 kHz offset from the carrier on trace 1 using the RPG knob

Press **Return**, **Select Marker**, **Marker 2**, **Normal**
Press **Properties**, **Select Marker**, **Marker 2**, **Marker Trace**, **Trace 3**
Press **Relative To**, **Marker 1**

Adjust Marker 2 to approximately 200 kHz offset from the carrier on trace 3 using the RPG knob

Press **Marker Function**, **Select Marker**, **Marker 1**, **Marker Noise**
Press **Select Marker**, **Marker 2**, **Marker Noise**

Figure 8-16 Improved Phase Noise Measurement with Delta Noise Markers



9 Measuring Digital Communications Signals

Introduction

The MXA Signal Analyzer makes power measurements on digital communication signals fast and repeatable by providing a comprehensive suite of power-based one-button automated measurements with pre-set standards-based format setups. The automated measurements also include pass/fail functionality that allow the user to quickly check if the signal passed the measurement.

Channel Power Measurements

This section explains how to make a channel power measurement on a W-CDMA (3GPP) mobile station. (A signal generator is used to simulate a base station.) This test measures the total RF power present in the channel. The results are displayed graphically as well as in total power (dB) and power spectral density (dBm/Hz).

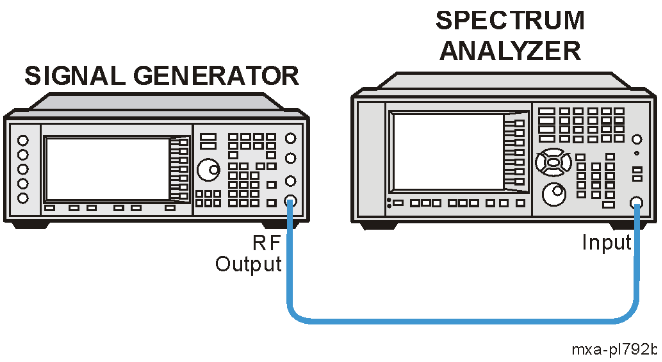
Measurement Procedure

Step 1. Setup the signal sources as follows:

Set the mode to W-CDMA
Set the frequency of the signal source to 1,920 MHz (Channel Number: $5 \times 1{,}920 = 9{,}600$).
Set the source amplitude to –20 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-1 **Setup for Obtaining Channel Power Measurement**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the radio standard and to toggle the device to mobile station:

Press **Mode Setup**, **Radio Std**, **3GPP W-CDMA**, **3GPP W-CDMA**, **Device** (MS).

Step 6. Set the center frequency to 1.920 GHz:

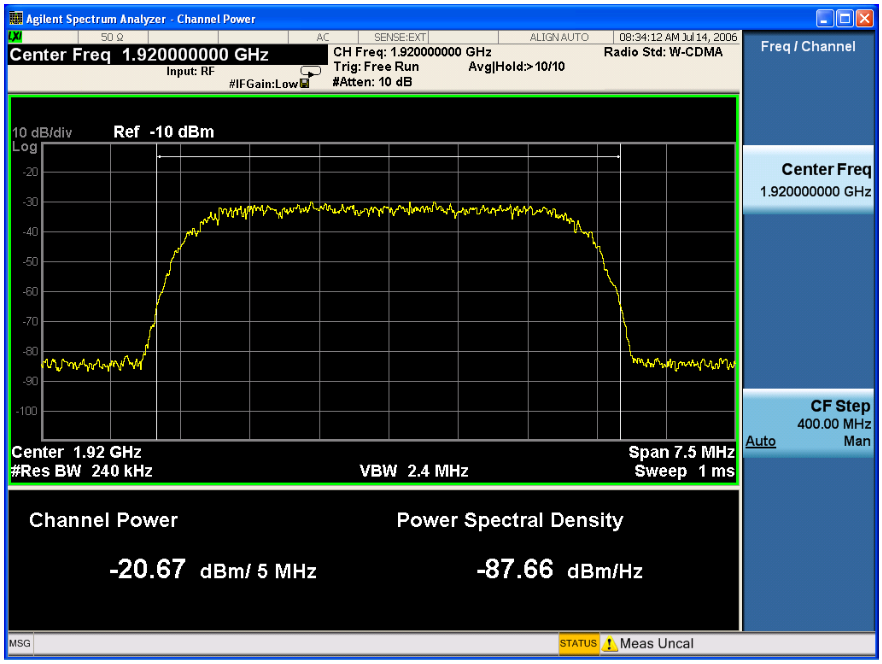
Press **FREQ Channel**, 1.920, **GHz**.

Step 7. Initiate the channel power measurement:

Press **MEAS**, **Channel Power**.

The `Channel Power` measurement result should look like Figure 9-2. The graph window and the text window showing the absolute power and its mean power spectral density values over 5 MHz are displayed.

Figure 9-2 Channel Power Measurement Result



Step 8. To determine what keys are available to change the measurement parameters from their default condition.

Press **Meas Setup**.

Occupied Bandwidth Measurements

This section explains how to make the occupied bandwidth measurement on a W-CDMA (3GPP) mobile station. (A signal generator is used to simulate a base station.) The instrument measures power across the band, and then calculates its 99.0% power bandwidth.

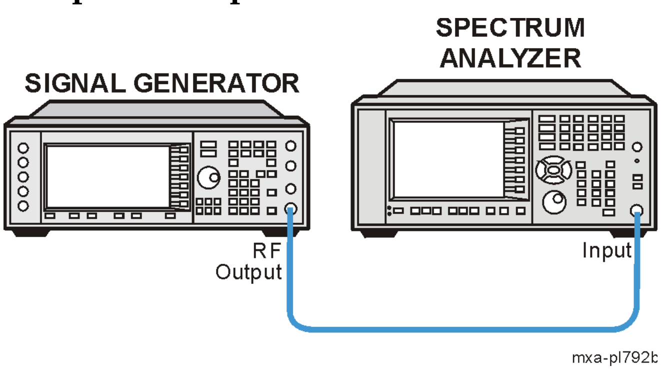
Measurement Procedure

Step 1. Setup the signal sources as follows:

Set the mode to W-CDMA
Set the frequency of the signal source to 1,920 MHz (Channel Number: $5 \times 1,920 = 9,600$).
Set the source amplitude to –20 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-3 **Setup for Occupied Bandwidth Measurement**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the radio standard and to toggle the device to mobile station:

Press **Mode Setup**, **Radio Std**, **3GPP W-CDMA**, **3GPP W-CDMA**, **Device** (MS).

Step 6. Set the center frequency to 1.920 GHz:

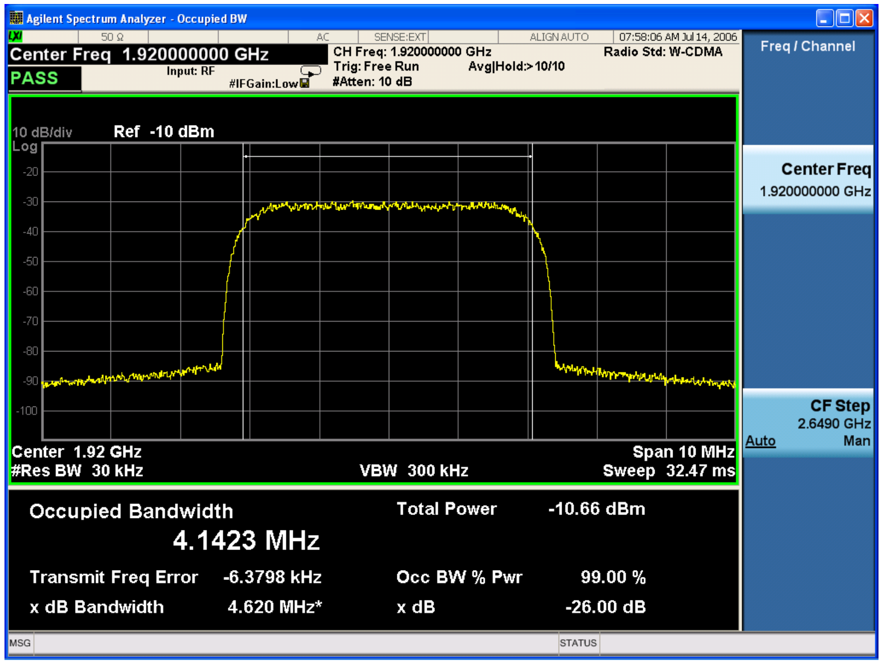
Press **FREQ Channel**, **Center Freq**, 1.920, **GHz**.

Step 7. Initiate the occupied bandwidth measurement:

Press **MEAS**, **Occupied BW**.

The `Occupied BW` measurement result should look like the Figure 9-4.

Figure 9-4 **Occupied Bandwidth Measurement Result**



Troubleshooting Hints

Any distortion such as harmonics or intermodulation, for example, produces undesirable power outside the specified bandwidth.

Shoulders on either side of the spectrum shape indicate spectral regrowth and intermodulation. Rounding or sloping of the top shape can indicate filter shape problems.

Making Adjacent Channel Power (ACP) Measurements

The adjacent channel power (ACP) measurement is also referred to as the adjacent channel power ratio (ACPR) and adjacent channel leakage ratio (ACLR). We use the term ACP to refer to this measurement.

ACP measures the total power (rms voltage) in the specified channel and up to six pairs of offset frequencies. The measurement result reports the ratios of the offset powers to the main channel power.

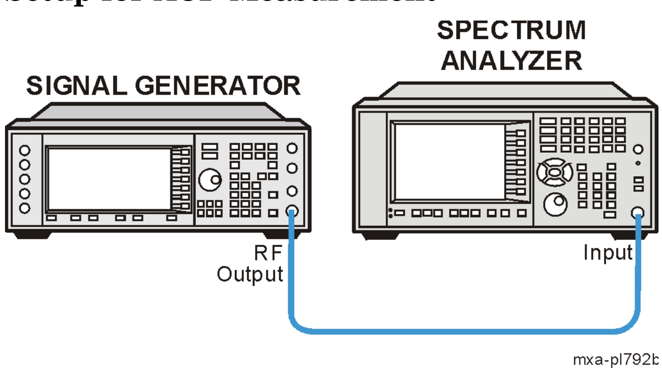
The following example shows how to make an ACP measurement on a W-CDMA base station signal broadcasting at 1.96 GHz. (A signal generator is used to simulate a base station.)

Step 1. Setup the signal sources as follows:

Setup a W-CDMA signal
Set the source frequency to 1.96 GHz.
Set the source amplitudes to –10 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-5 Setup for ACP Measurement



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency to 1.96 GHz:

Press **FREQ Channel**, **Center Freq**, 1.96, **GHz**.

Step 6. Set the analyzer radio mode to W-CDMA as a base station device:

Press **Mode Setup**, **Radio Std**, **3GPP W-CDMA**.
Press **Mode Setup**, **Radio Std Setup**, **Device** (BTS).

Step 7. Select the adjacent channel power one-button measurement from the measure menu and then optimize the attenuation setting suitable for the ACP measurement (see Figure 9-6):

Press **Meas**, **ACP**.
Press **Meas Setup**, **AMPTD**, **Attenuation**, **Adjust Atten for Min Clip**.

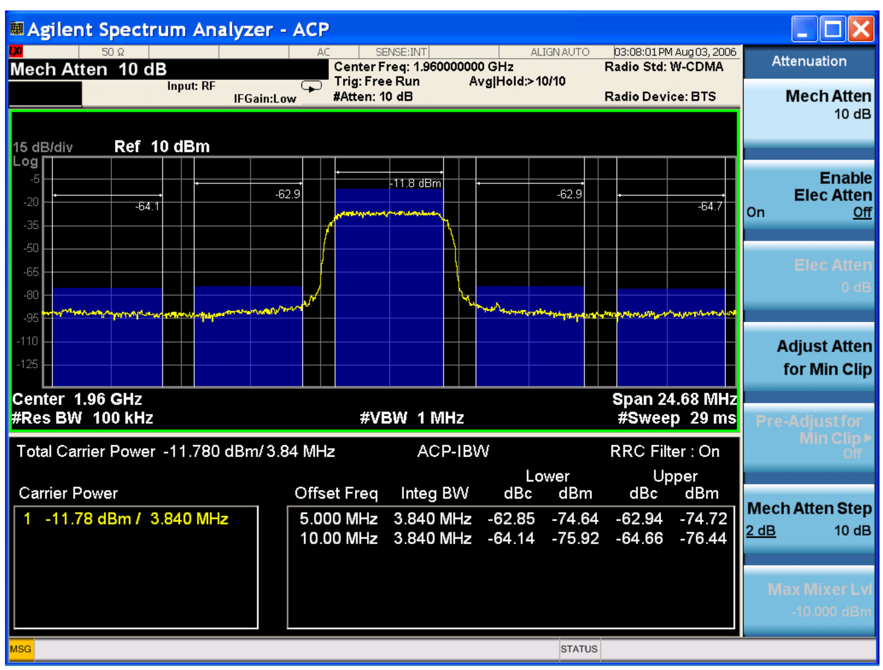
NOTE **Adjust Atten for Min Clip** protects against input signal overloads, but does not necessarily set the input attenuation and reference level for optimum measurement dynamic range.

To improve the measurement repeatability, increase the sweep time to smooth out the trace (average detector must be selected). Measurement repeatability can be traded off with sweep time.

Step 8. To increase dynamic range, **Noise Correction** can be used to factor out the added power of the noise floor effects. Noise correction is very useful when measuring signals near the noise floor of the analyzer.

Press **Meas Setup**, **More**, **More**, **Noise Correction** (On).

Figure 9-6 ACP Measurement on a Base Station W-CDMA Signal



The frequency offsets, channel integration bandwidths, and span settings can all be modified from the default settings selected by the radio standard.

Two vertical white lines, in the center of the screen, indicate the bandwidth limits of the central channel being measured.

Offsets A and B are designated by the adjacent pairs of white lines, in this case: 5 MHz and 10 MHz from the center frequency respectively.

Step 9. View the results using the full screen:

Press **Display**, **Full Screen**.

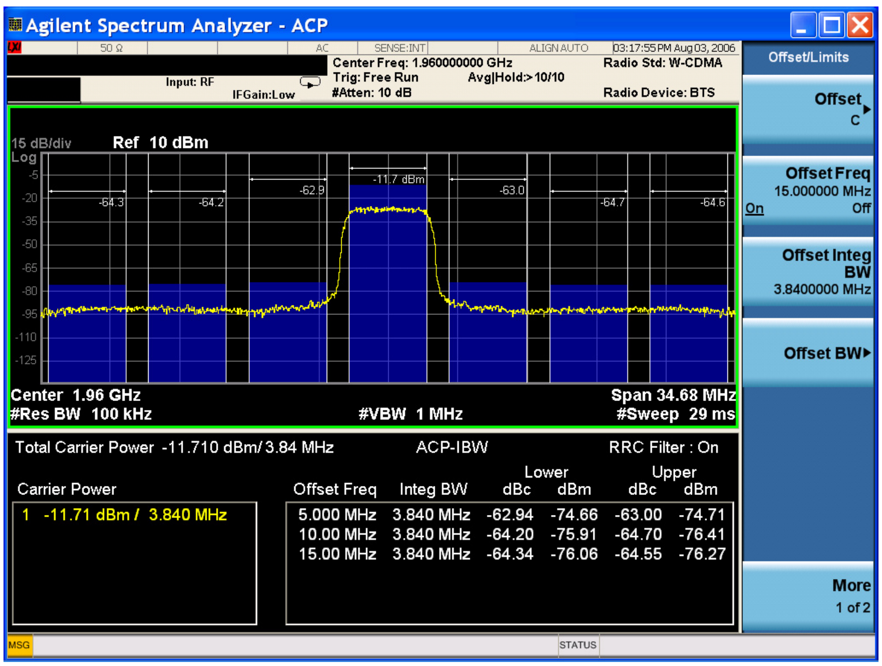
NOTE Press the **Full Screen** key again to exit the full screen display without changing any parameter values.

Step 10. Define a new third pair of offset frequencies:

Press **Meas Setup**, **Offset/Limits**, **Offset**, **C**, **Offset Freq** (On), 15, **MHz**.

This third pair of offset frequencies is offset by 15.0 MHz from the center frequency (the outside offset pair) as shown in Figure 9-7. Three further pairs of offset frequencies (D, E and F) are also available.

Figure 9-7 Measuring a Third Adjacent Channel



Step 11. Set pass/fail limits for each offset:

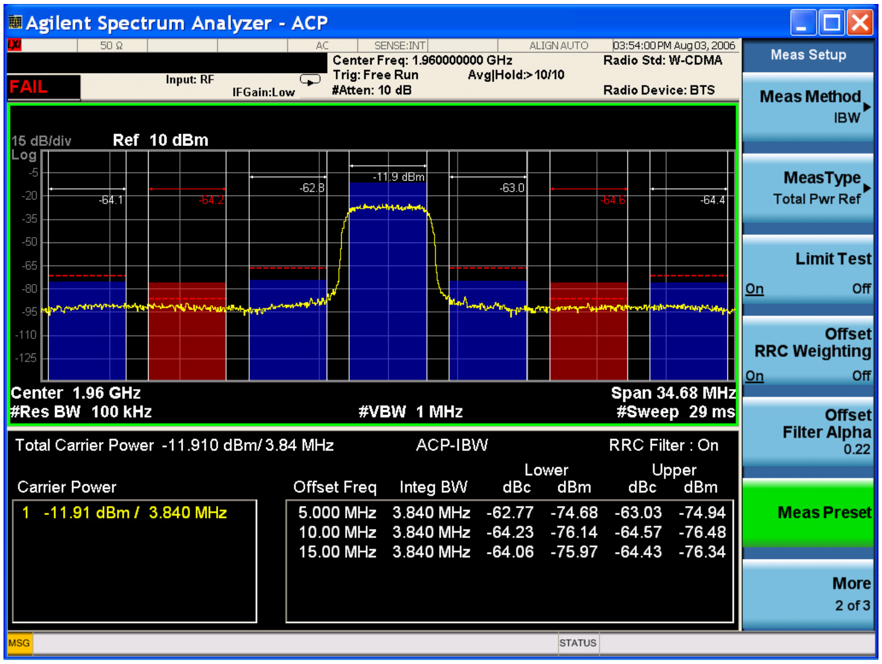
Press **Meas Setup**, **Offset/Limits**, **Offset**, **A**, **More**, **Rel Limit (Car)**, –55, **dB**, **Offset**, **B**, **Rel Limit (Car)**, –75, **dB**, **Offset**, **C**, **Rel Limit (Car)**, –60, **dB**.

Step 12. Turn the limit test on:

Press **Meas Setup**, **More**, **Limit Test** (On).

In Figure 9-8 notice that offsets A and C have passed, however offset B has failed. Power levels that fall above our specified `–75 dB for offset B, fail. The offset bar graph and the associated power level value are shaded red to identify a failure. The offset limits are shown as dashed lines.

Figure 9-8 Setting Offset Limits



NOTE You may increase the repeatability by increasing the sweep time.

Making Statistical Power Measurements (CCDF)

Complementary cumulative distribution function (CCDF) curves characterize a signal by providing information about how much time the signal spends at or above a given power level. The CCDF measurement shows the percentage of time a signal spends at a particular power level. Percentage is on the vertical axis and power (in dB) is on the horizontal axis.

All CDMA signals, and W-CDMA signals in particular, are characterized by high power peaks that occur infrequently. It is important that these peaks are preserved otherwise separate data channels can not be received properly. Too many peak signals can also cause spectral regrowth. If a CDMA system works well most of the time and only fails occasionally, this can often be caused by compression of the higher peak signals.

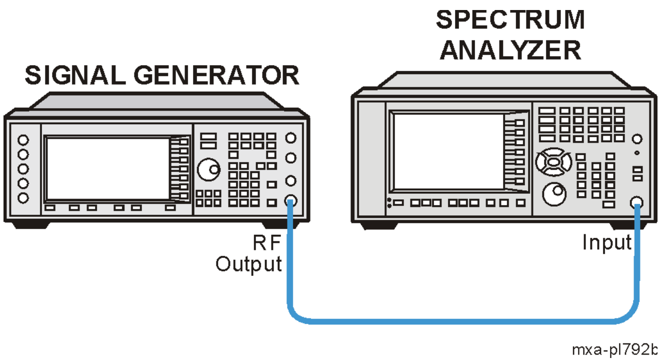
The following example shows how to make a CCDF measurement on a W-CDMA signal broadcasting at 1.96 GHz. (A signal generator is used to simulate a base station.)

Step 1. Setup the signal sources as follows:

Setup a W-CDMA down link signal
Set the source frequency to 1.96 GHz.
Set the source amplitudes to –10 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-9 **Setup for CCDF Measurement**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency to 1.96 GHz:

Press **FREQ Channel**, **Center Freq**, 1.96, **GHz**.

Step 6. Set the analyzer radio mode to W-CDMA as a base station device:

Press **Mode Setup**, **Radio Std**, **3GPP W-CDMA**, **3GPP W-CDMA, Device** (BTS).

Step 7. Select the CCDF one-button measurement from the measure menu and then optimize the attenuation level and attenuation settings suitable for the CCDF measurement:

Press **Meas**, **Power Stat CCDF**.
Press **AMPTD**, **Attenuation**, **Adjust Atten for Min Clip**.

Figure 9-10 Power Statistics CCDF Measurement on a W-CDMA Signal

Step 8. Store your current measurement trace for future reference:

Press **Trace/Detector**, **Store Ref Trace**.

When the power stat CCDF measurement is first made, the graphical display should show a signal typical of pure noise. This is labelled Gaussian, and is shown in aqua. Your CCDF measurement is displayed as a yellow plot. You have stored this measurement's plot to make for easy comparison with subsequent measurements. Refer to Figure 9-10.

Step 9. Display the stored trace:

Press **Trace/Detector**, **Ref Trace** (On).

Step 10. Change the measurement bandwidth to 1 MHz:

Press **Meas Setup**, **Info BW**, 1, **MHz**.

The stored trace from your last measurement is displayed as a magenta plot (as shown in Figure 9-11), and allows direct comparison with your current measurement (yellow trace).

Figure 9-11 Storing and Displaying a Power Stat CCDF Measurement

NOTE	If you choose a measurement bandwidth setting that the analyzer cannot display, it automatically sets itself to the closest available bandwidth setting.

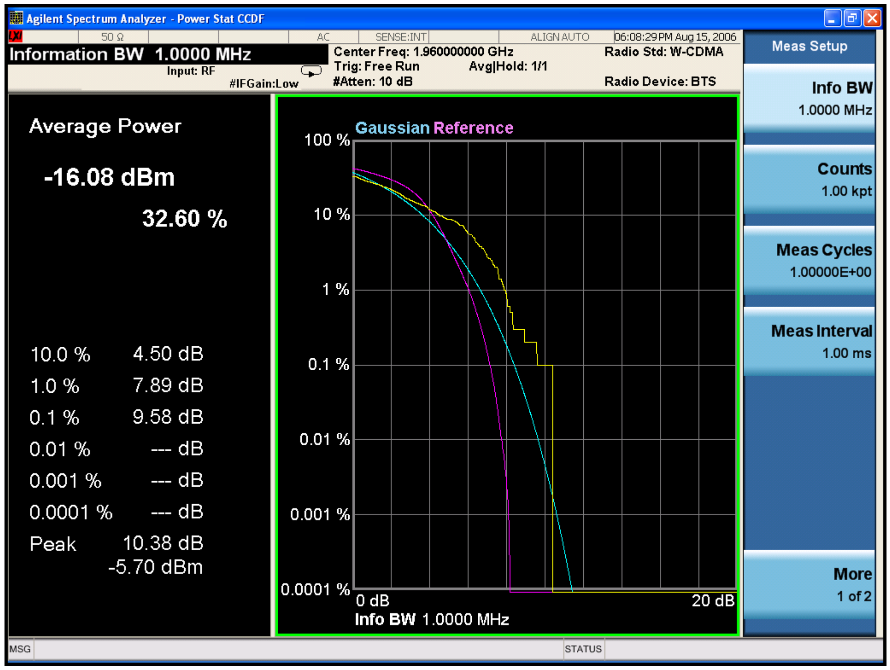
Step 11. Change the number of measured points from 10,000,000 (10.0Mpt) to 1,000 (1kpt):

Press **Meas Setup**, **Counts**, 1, **kpt**.

Reducing the number of points decreases the measurement time, however the number of points is a factor in determining measurement uncertainty and repeatability. Notice how the displayed plot loses a lot of its smoothness. You are gaining speed but reducing repeatability and increasing measurement uncertainty. refer to Figure 9-12.

NOTE	The number of points collected per sweep is dependent on the sampling rate and the measurement interval. The number of samples that have been processed are indicated at the top of the screen. The graphical plot is continuously updated so you can see it getting smoother as measurement uncertainty is reduced and repeatability improves.

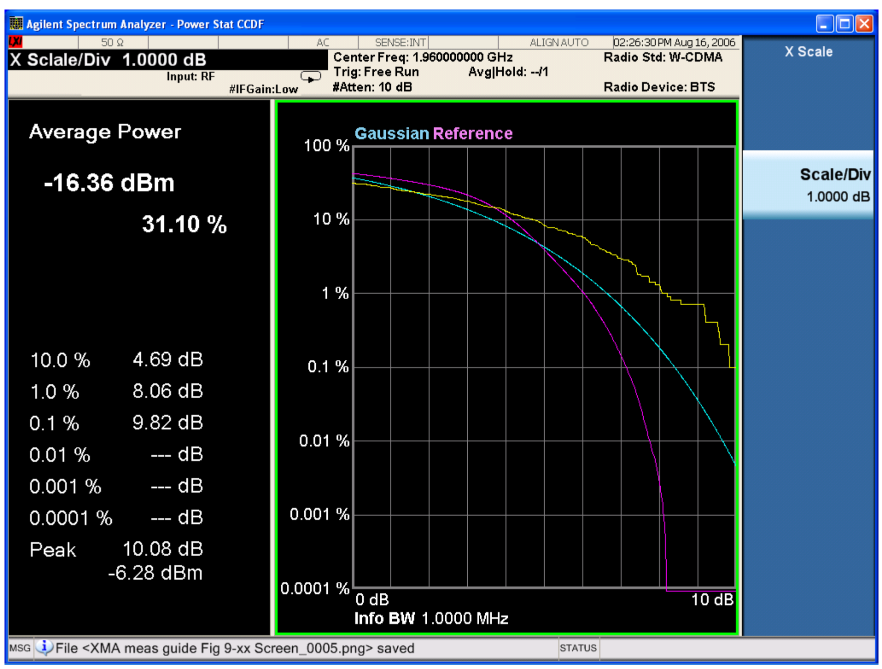
Figure 9-12 Reducing the Measurement Points to 1 kpt



Step 12. Change the scaling of the X-axis to 1 dB per division to optimize your particular measurement:

Press **SPAN X Scale**, **Scale/Div**, 1, **dB**. Refer to Figure 9-13.

Figure 9-13 Reducing the X Scale to 1 dB



Making Burst Power Measurements

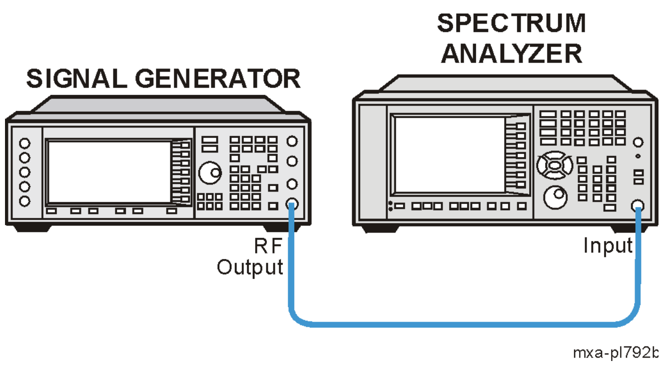
The following example demonstrates how to make a burst power measurement on a Bluetooth™ signal broadcasting at 2.402 GHz. (A signal generator is used to simulate a Bluetooth™ signal.)

Step 1. Setup the signal sources as follows:

Setup a Bluetooth™ signal transmitting DH1 packets
Set the source frequency to 2.402 GHz.
Set the source amplitudes to –10 dBm.
Set Burst on.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-14 Setup for Burst Power Measurement



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency to 2.402 GHz:

Press **FREQ Channel**, **Center Freq**, 2.402, **GHz**.

Step 6. Set the analyzer radio mode to Bluetooth™ and check to make sure packet type DH1 is selected:

Press **Mode Setup**, **Radio Std**, **More**, **Bluetooth**, **Bluetooth**, **DH1**.

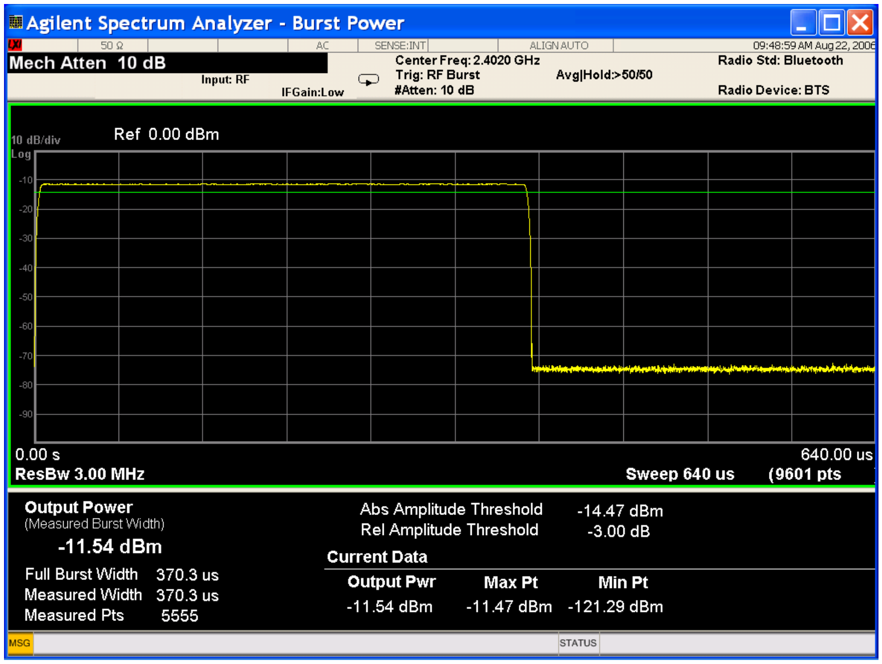
Step 7. Select the burst power one-button measurement from the measure menu and optimize the attenuation level:

Press **Meas**, **Burst Power**.
Press **AMPTD**, **Attenuation**, **Adjust Atten for Min Clip**.

Step 8. View the results of the burst power measurement using the full screen (See Figure 9-15):

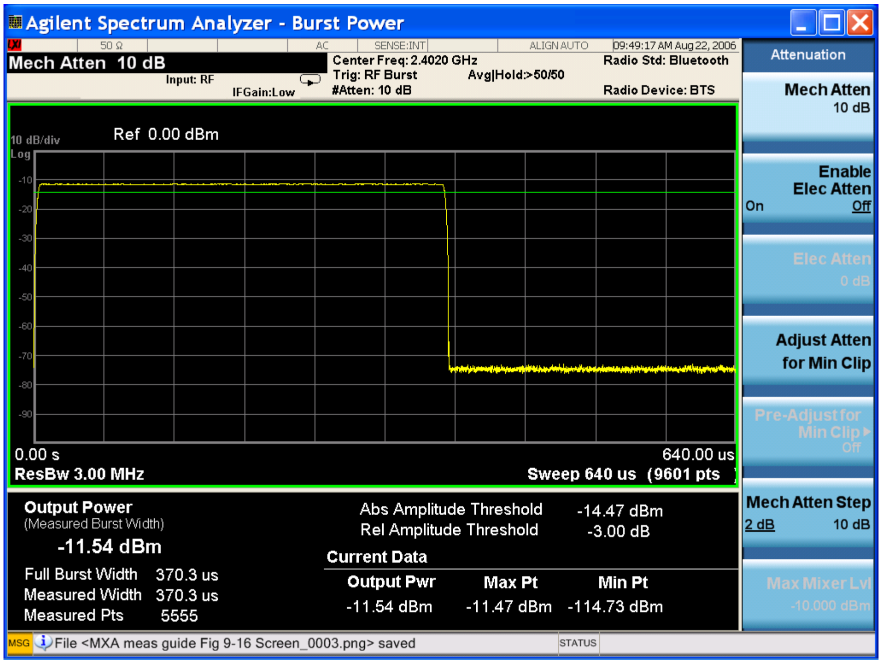
Press **View/Display**, **Full Screen**.

Figure 9-15 Full Screen Display of Burst Power Measurement Results



NOTE Press the **Full Screen** key again to exit the full screen display without changing any parameter values. Refer to Figure 9-16.

Figure 9-16 Normal Screen Display of Burst Power Measurement Results



Step 9. Select one of the following three trigger methods to capture the bursted signal (RF burst is recommended, if available):

Press **Trigger**, **RF Burst**.

For more information on trigger selections see “Trigger Concepts” on page 142.

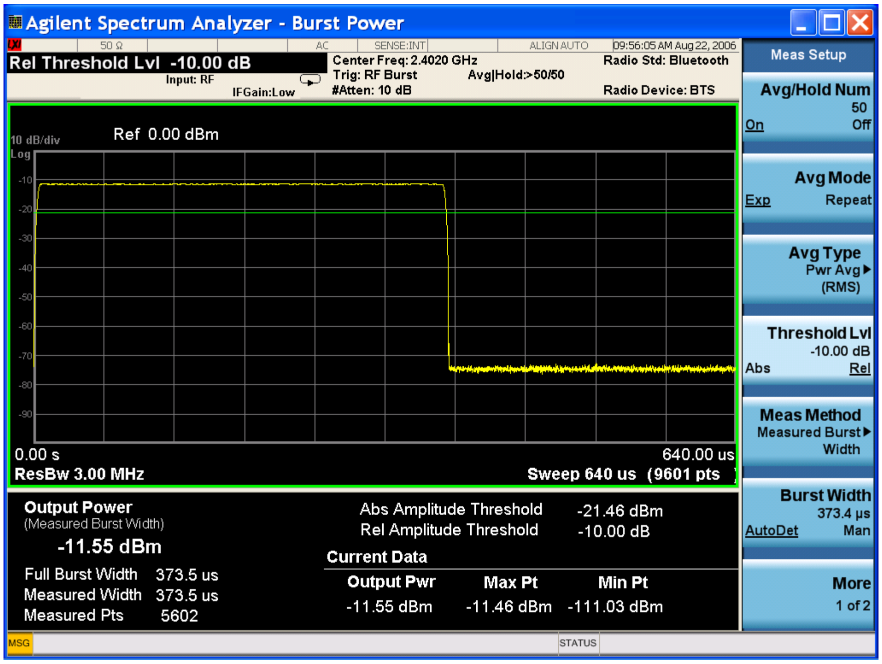
NOTE Although the trigger level allows the analyzer to detect the presence of a burst, the time samples contributing to the burst power measurement are determined by the threshold level, as described next.

Step 10. Set the relative threshold level above which the burst power measurement is calculated:

Press **Meas Setup**, **Threshold Lvl** (Rel), –10, **dB**.

The burst power measurement includes all points above the threshold and no points below. The threshold level is indicated on the display by the green horizontal line (for video triggering it is the upper line). In this example, the threshold level has been set to be 10 dB below the relative level of the burst. The mean power of the burst is measured from all data above the threshold level. Refer to Figure 9-17.

Figure 9-17 Burst Power Measurement Results with Threshold Level Set

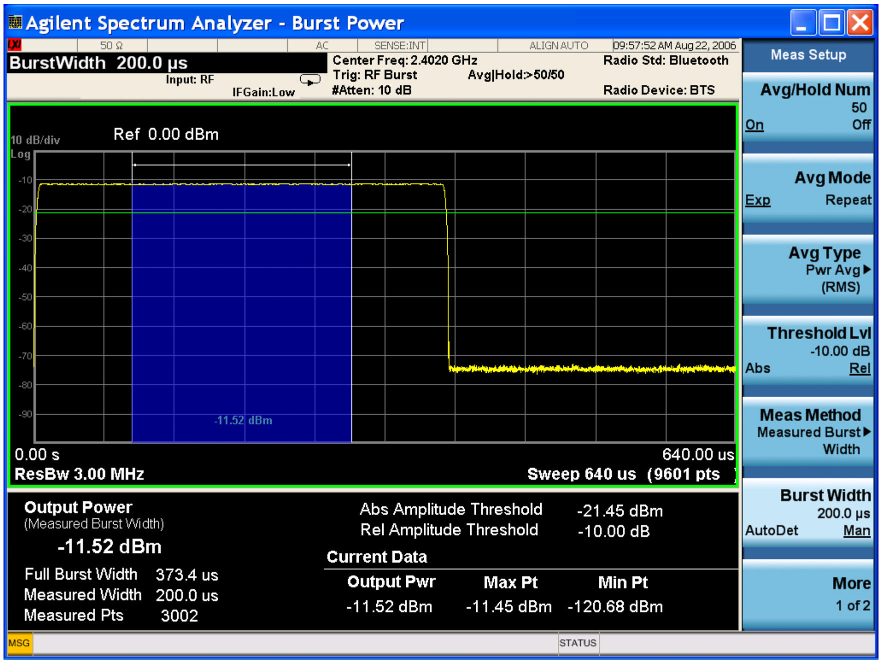


Step 11. Set the burst width to measure the central 200 µs of the burst and enable bar graph:

Press **View/Display**, **Bar Graph** (On)
Press **Meas Setup**, **Meas Method**, **Measured Burst Width**, **Burst Width** (Man), 200, µ**s**.

The burst width is indicated on the screen by two vertical white lines and a blue power bar. Manually setting the burst width allows you to make it a long time interval (to include the rising and falling edges of the burst) or to make it a short time interval, measuring a small central section of the burst. Refer to Figure 9-18.

Figure 9-18 Bar Graph Results with Measured Burst Width Set



NOTE If you set the burst width manually to be wider than the screen's display, the vertical white lines move off the edges of the screen. This could give misleading results as only the data on the screen can be measured.

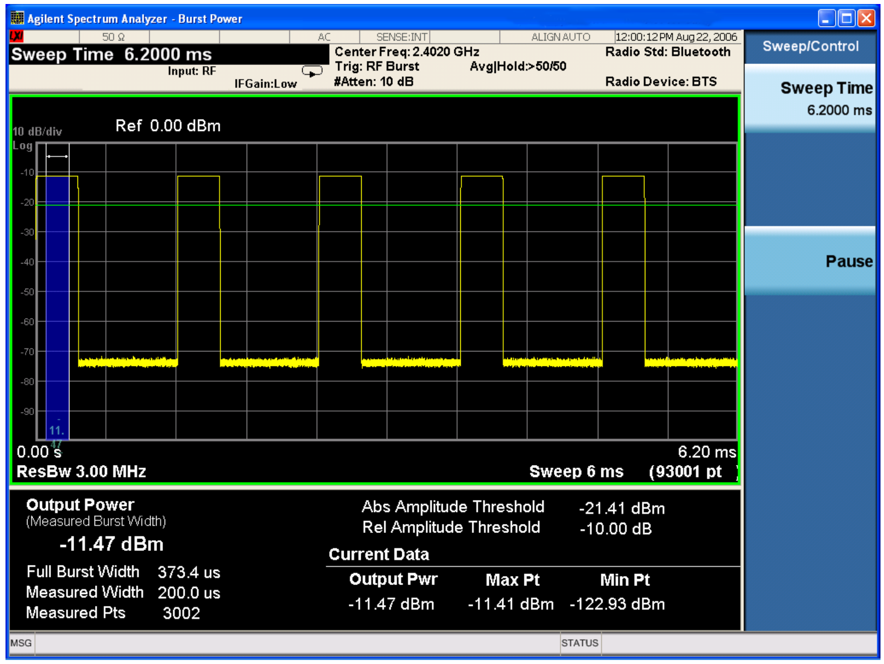
NOTE The Bluetooth™ standard states that power measurements should be taken over at least 20% to 80% of the duration of the burst.

Step 12. Increase the sweep time to display more than one burst at a time:

Press **Sweep/Control**, **Sweep Time**, 6200, µ**s** (or 6.2, **ms**).

The screen display shows several bursts in a single sweep as in Figure 9-19. The burst power measurement measures the mean power of the first burst, indicated by the vertical white lines and blue power bar.

Figure 9-19 Displaying Multiple Bursts



NOTE Although the burst power measurement still runs correctly when several bursts are displayed simultaneously, the timing accuracy of the measurement is degraded. For the best results (including the best trade-off between measurement variations and averaging time), it is recommended that the measurement be performed on a single burst.

Spurious Emissions Measurements

The following example demonstrates how to make a spurious emissions measurement on a multitone signal used to simulate a spurious emission in a measured spectrum.

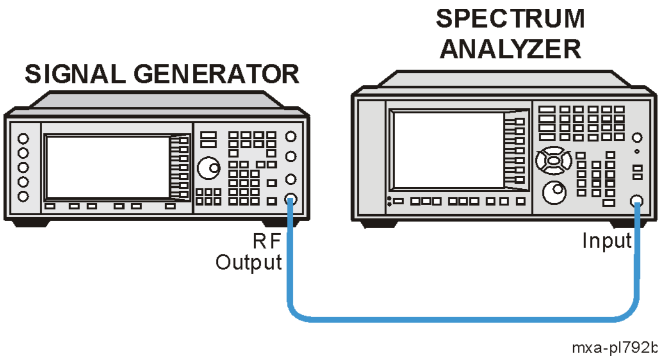
Measurement Procedure

Step 1. Setup the signal sources as follows:

Setup a multitone signal with 8 tones with a 2.0 MHz frequency spacing.
Set the source frequency to 1.950 GHz.
Set the source amplitudes to –50 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-20 **Setup for Spurious Emissions Measurement**



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency to 1.950 GHz:

Press **FREQ Channel**, **Center Freq**, 1.950, **GHz**.

Step 6. Select the spurious emissions one-button measurement from the measure menu:

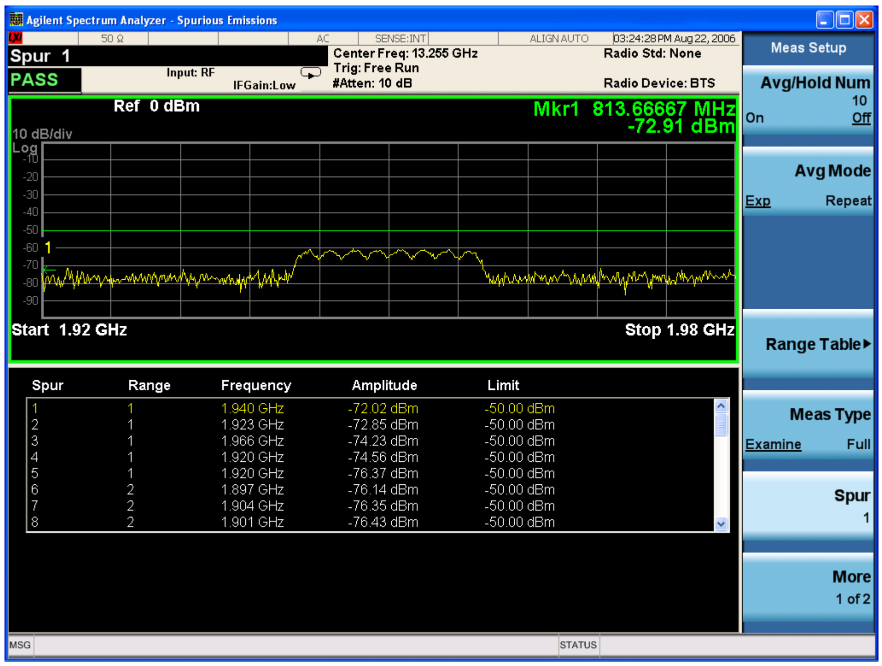
Press **Meas**, **More**, **Spurious Emissions**.

Step 7. You may Focus the display on a specific spurious emissions signal:

Press **MEAS Setup**, **Spur**, 1, Enter (or enter the number of the spur of interest).
Press **Meas Type** to highlight **Examine**.

The `Spurious Emission` result should look like Figure 9-21. The graph window and a text window are displayed. The text window shows the list of detected spurs. Each line item includes the spur number, the range in which the spur was detected, the power of the spur, and the limit value against which the spur amplitude is tested.

Figure 9-21 Spurious Emission Measurement Result



Step 8. You may customize the tested ranges for spurious emissions (initially 5 default ranges and parameters are loaded into the range table):

Press **MEAS Setup**, **Range Table**, then select and edit the available parameters.

Troubleshooting Hints

Spurious emissions measurements can reveal the presence of degraded or defective parts in the transmitter section of the UUT. The following are examples of problems which, once indicated by testing, may require further attention:

- Faulty DC power supply control of the transmitter power amplifier
- RF power controller of the pre-power amplifier stage
- I/Q control of the baseband stage
- Reduction in the gain and output power level of the amplifier due to

a degraded gain control and/or increased distortion

- Degradation of amplifier linearity and other performance characteristics

Power amplifiers are one of the final stage elements of a base transmitter and play a critical part in meeting the important power and spectral efficiency specifications. Measuring the spectral response of these amplifiers to complex wideband signals is crucial to linking amplifier linearity and other performance characteristics to the stringent system specifications.

Spectrum Emission Mask Measurements

This section explains how to make the spectrum emission mask measurement on a W-CDMA (3GPP) mobile station. (A signal generator is used to simulate a base station.) SEM compares the total power level within the defined carrier bandwidth and the given offset channels on both sides of the carrier frequency, to levels allowed by the standard. Results of the measurement of each offset segment can be viewed separately.

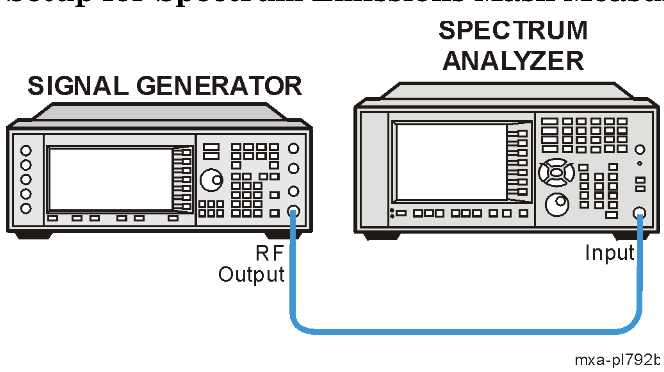
Measurement Procedure

Step 1. Setup the signal sources as follows:

Setup a W-CDMA uplink signal
Set the source frequency to 1,920 MHz (Channel Number: 5 × 1,920 = 9,600).
Set the source amplitudes to 0 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 9-5.

Figure 9-22 Setup for Spectrum Emissions Mask Measurement



Step 3. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements:

Press **MODE**, **Spectrum Analyzer**.

Step 4. Preset the analyzer:

Press **Mode Preset**.

Step 5. Set the analyzer center frequency to 1.920 GHz:

Press **FREQ Channel**, **Center Freq**, 1.920, **GHz**.

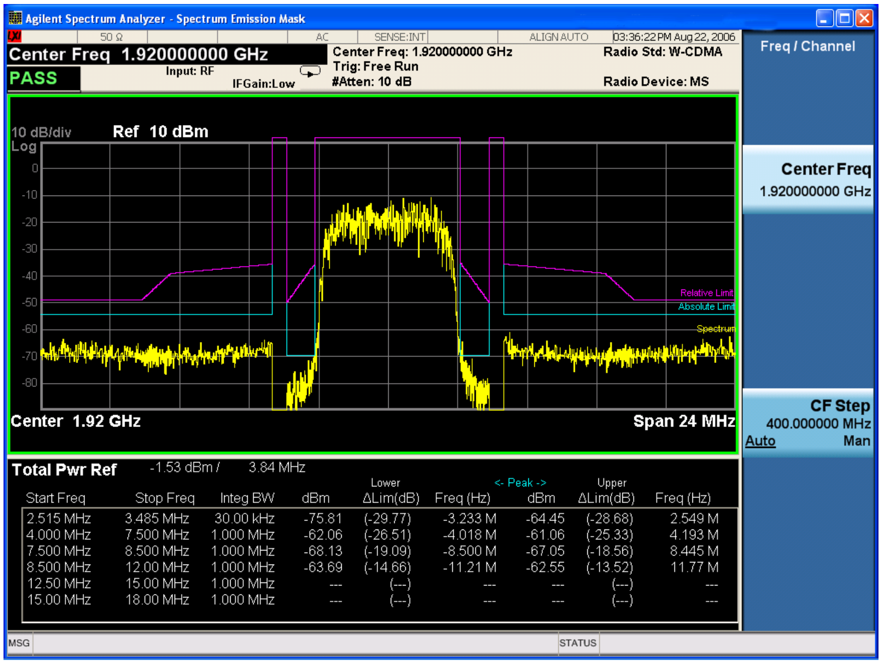
Step 6. Set the analyzer radio mode to W-CDMA as a mobile station device:

Press **Mode Setup**, **Radio Std**, **3GPP W-CDMA**, **3GPP W-CDMA, Device** (MS).

Step 7. Initiate the spectrum emission mask measurement.

Press **MEAS**, **More**, **Spectrum Emission Mask**.

Figure 9-23 Spectrum Emission Mask Measurement Result - (Default) View



The Spectrum Emission Mask measurement result should look like Figure 9-23. The text window shows the reference total power and the absolute peak power levels which correspond to the frequency bands on both sides of the reference channel.

Troubleshooting Hints

This spectrum emission mask measurement can reveal degraded or defective parts in the transmitter section of the UUT. The following examples are those areas to be checked further.

- Faulty DC power supply control of the transmitter power amplifier.
- RF power controller of the pre-power amplifier stage.
- I/Q control of the baseband stage.
- Some degradation in the gain and output power level of the amplifier due to the degraded gain control and/or increased distortion.
- Some degradation of the amplifier linearity or other performance characteristics.

Power amplifiers are one of the final stage elements of a base or mobile transmitter and are a critical part of meeting the important power and spectral efficiency specifications. Since spectrum emission mask measures the spectral response of the amplifier to a complex wideband signal, it is a key measurement linking amplifier linearity and other performance characteristics to the stringent system specifications.

10 Demodulating AM and FM Signals

Measuring the Modulation Rate of an AM Signal

This section demonstrates how to determine parameters of an AM signal, such as modulation rate and modulation index (depth) by using frequency and time domain measurements (see the concepts chapter "AM and FM Demodulation Concepts" on page 146 for more information).

To obtain an AM signal, you can either connect a source transmitting an AM signal, or connect an antenna to the analyzer input and tune to a commercial AM broadcast station. For this demonstration an RF source is used to emulate an AM signal.

Step 1. Connect an Agilent ESG RF signal source to the analyzer RF INPUT. Setup the signal sources as follows:

Set the source frequency to 300 MHz.
Set the source amplitudes to –10 dBm.
Set the AM depth to 80%.
Set the AM rate to 1 kHz.
Turn AM on.

Step 2. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 3. Preset the analyzer:

Press **Mode Preset**.

Step 4. Set the center frequency, span, RBW and the sweep time:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 500, **kHz**.
Press **BW**, **Res BW**, 30, **kHz**.
Press **Sweep/Control**, **Sweep Time**, 20, **ms**.

Step 5. Change the y-scale type to linear:

Press **AMPTD Y Scale**, **Scale Type** (Lin).
The y-axis units will automatically set to volts

Step 6. Position the signal peak near the first graticule below the reference level:

Press **AMPTD Y Scale**, **Ref Level**, (rotate front-panel knob).

Step 7. Set the analyzer in zero span to make time-domain measurements:

Press **SPAN X Scale**, **Zero Span**.
Press **Sweep/Control**, **Sweep Time**, 5, **ms**.

Step 8. Use the video trigger to stabilize the trace:

Press **Trigger**, **Video**.

Since the modulation is a steady tone, you can use video trigger to trigger the analyzer sweep on the waveform and stabilize the trace, much like an oscilloscope. See Figure 10-1.

NOTE If the trigger level is set too high or too low when video trigger mode is activated, the sweep stops. You need to adjust the trigger level up or down with the front-panel knob until the sweep begins again.

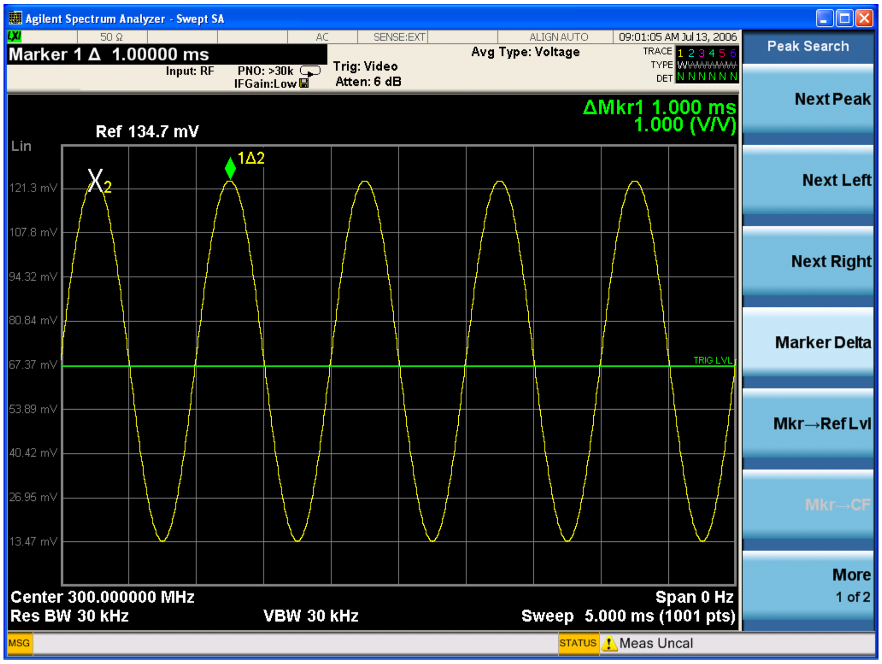
Press **Trigger**, **Video**, **Trigger Level**, (rotate front-panel knob).

Step 9. Measure the AM rate using delta markers:

Press **Peak Search**, **Marker Delta**, **Next Right** or **Next Left**.

Use markers and delta markers to measure the AM rate. Place the marker on a peak and then use a delta marker to measure the time difference between the peaks (this is the AM rate of the signal)

Figure 10-1 Measuring Time Parameters



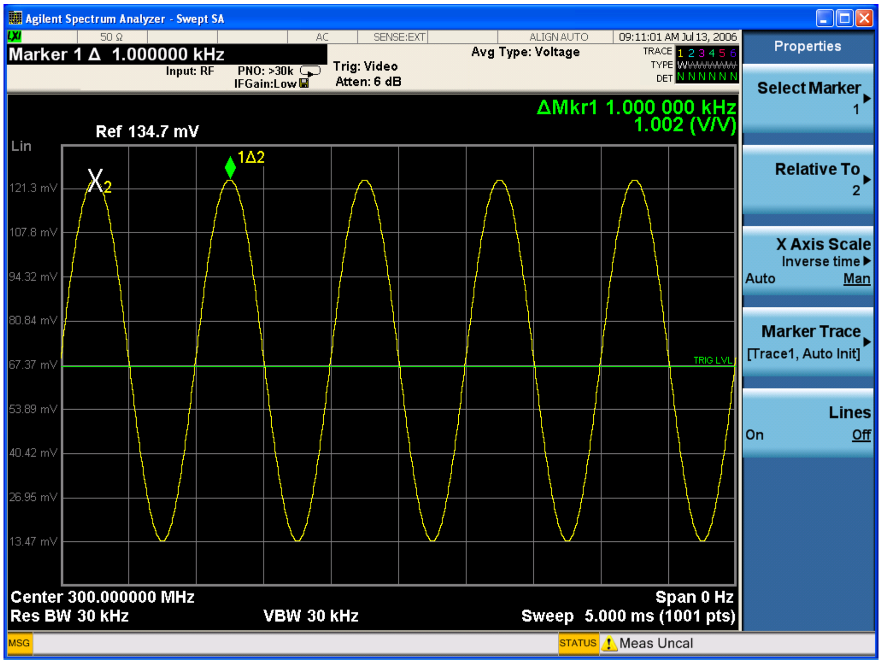
NOTE Make sure the delta markers above are placed on adjacent peaks. See Figure 10-1. The frequency or the AM rate is 1 divided by the time between adjacent peaks:

AM Rate = 1/1.0 ms = 1 kHz

The signal analyzer can also make this rate calculation by changing the marker readout to inverse time.

Press **Marker**, **Properties**, **X Axis Scale**, **Inverse Time**.

Figure 10-2 Measuring Time Parameters with Inverse Time Readout



Another way to calculate the modulation rate would be to view the signal in the frequency domain and measure the delta frequency between the peak of the carrier and the first sideband.

Measuring the Modulation Index of an AM Signal

This procedure demonstrates how to use the signal analyzer as a fixed-tuned (time-domain) receiver to measure the modulation index as a percent AM value of an AM signal.

Step 1. Connect an Agilent ESG RF signal source to the analyzer RF INPUT. Setup the signal sources as follows:

Set the source frequency to 300 MHz.
Set the source amplitudes to –10 dBm.
Set the AM depth to 80%.
Set the AM rate to 1 kHz.
Turn AM on.

Step 2. Set the analyzer to the Spectrum Analyzer mode and enable the spectrum analyzer measurements

Press **MODE**, **Spectrum Analyzer**.

Step 3. Preset the analyzer:

Press **Mode Preset**.

Step 4. Set the center frequency, span, RBW and the sweep time:

Press **FREQ Channel**, **Center Freq**, 300, **MHz**.
Press **SPAN X Scale**, **Span**, 500, **kHz**.
Press **BW**, **Res BW**, 30, **kHz**.
Press **Sweep/Control**, **Sweep Time**, 20, **ms**.

Step 5. Set the y-axis units to volts:

Press **AMPTD Y Scale**, **More**, **Y-Axis Units**, **V** (Volts).

Step 6. Position the signal peak near the reference level:

Press **AMPTD Y Scale**, **Ref Level**, (rotate front-panel knob).

Step 7. Change the y-scale type to linear:

Press **AMPTD Y Scale**, **Scale Type** (Lin).

Step 8. Set the analyzer in zero span to make time-domain measurements:

Press **SPAN X Scale**, **Zero Span**.
Press **Sweep/Control**, **Sweep Time**, 5, **ms**.

Step 9. Place the analyzer in free run trigger mode:

Press **Trigger**, **Free Run**.

Step 10. Increase the sweep time and decrease the VBW so that the waveform is displayed as a flat horizontal signal:

Press **Sweep/Control**, **Sweep Time**, 5, **s**.
Press **BW**, **Video BW**, 30, **Hz**.

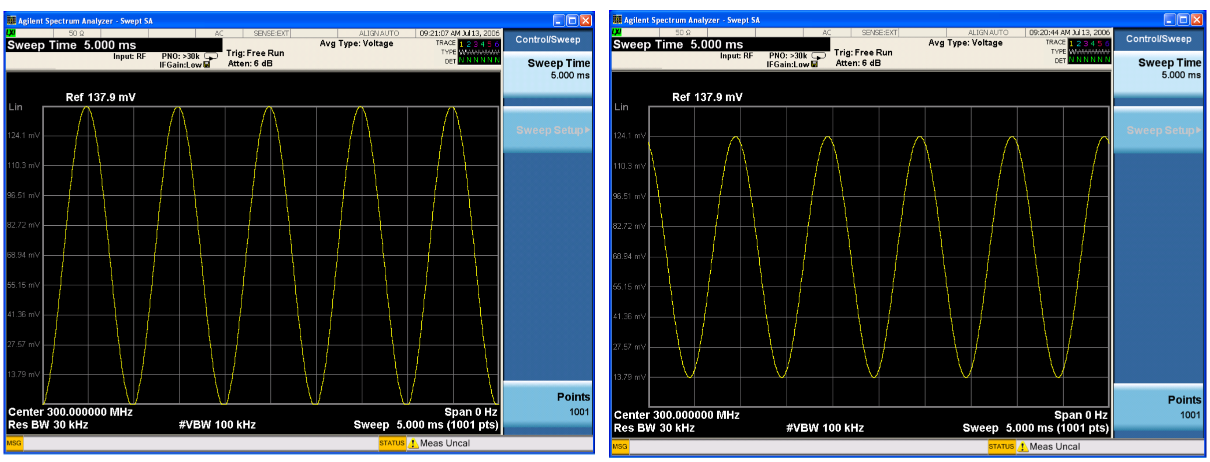
Step 11. Center the flat waveform at the mid-point of the y-axis and then widen the VBW and decrease the sweep time to display the waveform as a sine wave:

Press **AMPTD Y Scale**, **Ref Level**, (rotate front-panel knob).
Press **BW**, **Video BW**, 100, **kHz**.
Press **Sweep/Control**, **Sweep Time**, 5, **ms**.

Step 12. Measure the modulation index of the AM signal:

To measure the modulation index as % AM, read the trace as follows (see Figure 10-3 for display examples): 100% AM extends from the top graticule down to the bottom graticule. 80% AM (as in this example) is when the top of the signal is at 1 division below the top graticule and 1 division above the bottom graticule. To determine % AM of your signal count each y-axis division as 10%.

Figure 10-3 AM Signal Measured in the Time Domain



LEFT: 100% AM Signal (Modulation Index = 1)
RIGHT: 80% AM Signal (Modulation Index = 0.8)

11 IQ Signal Analysis

Capturing wideband signals for further analysis

This section demonstrates how to capture complex time domain data from wide bandwidth RF signals. This mode preserves the instantaneous vector relationships of time, frequency, phase and amplitude contained within the selected digitizer span or analysis BW, at the analyzer's center frequency, for output as IQ data. This IQ data can then be utilized internally or output over LAN, USB or GPIB for use with external analysis tools.

The standard 10 MHz Analysis BW and optional 25 MHz Analysis BW digitizers used to capture the wide bandwidth RF signals can be accessed from the front panel in IQ Analyzer (Basic) mode. This IQ Analyzer mode provides basic setup, RF (FFT based) and IQ analysis tools

Within the IQ Analyzer mode, basic frequency domain, time domain and IQ measurements are available as initial signal and data verification tools in preparation for deriving the IQ data output.

The Complex Spectrum measurement provides a display in the upper window of power versus frequency with current (yellow trace) and average (blue trace) data. In addition, an IQ waveform of voltage versus time is provided in the lower window.

The IQ Waveform measurement provides a time domain view of the RF signal envelope with power versus time or an IQ waveform of voltage versus time.

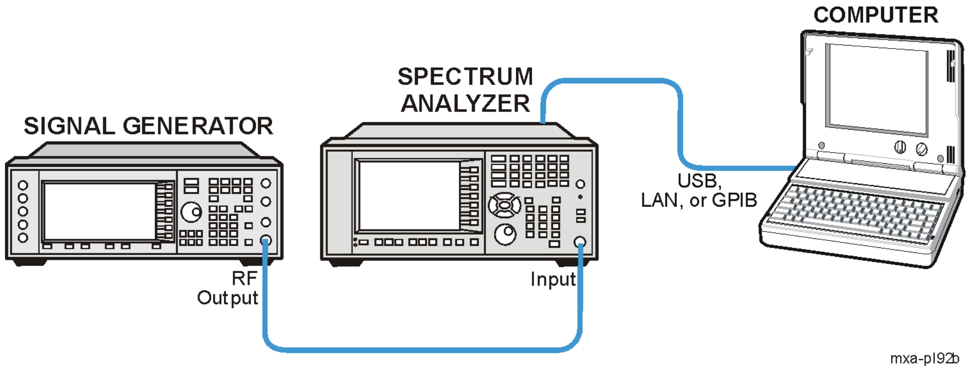
Measurement Procedure

Step 1. Setup the signal source as follows:

Set the mode to W-CDMA 3GPP with 4 carriers.
Set the frequency of the signal source to 1.0 GHz.
Set the source amplitude to -10 dBm.

Step 2. Connect the source RF OUTPUT to the analyzer RF INPUT as shown in Figure 11-1.

Figure 11-1 **Setup for** IQ Analysis



Step 3. Set the analyzer to IQ Analyzer mode and enable IQ data availability:

Press **Mode**, **IQ Analyzer (Basic)**

Step 4. Preset the analyzer mode:

Press **Mode Preset**

Step 5. Set the measurement center frequency:

Press **Freq Channel**, 1, **GHz**

Step 6. Set the measurement span/analysis bandwidth:

Press **Span X Scale**, 10, **MHz** (25 MHz if option B25 installed)

Step 7. Enable the Complex Spectrum measurement:

Press **Meas**, **Complex Spectrum** Refer to the default view in Figure 11-2 or Figure 11-3.

Figure 11-2 Spectrum and I/Q Waveform (Span 10 MHz)

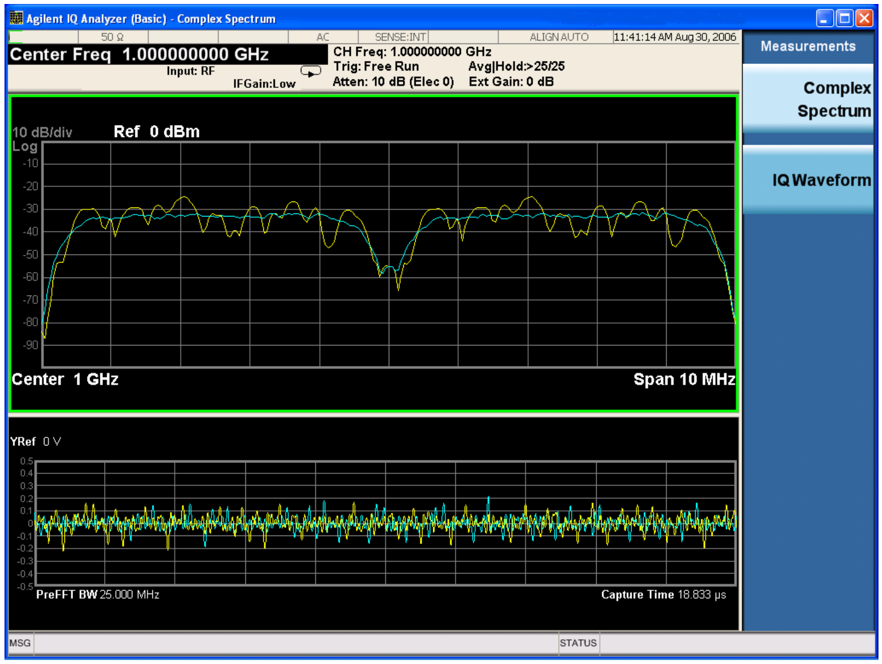
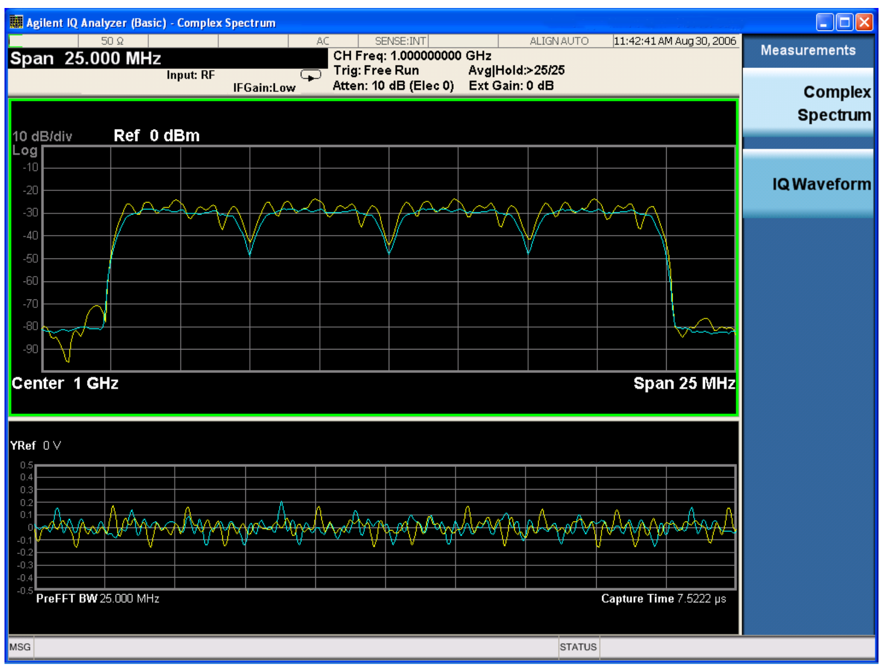


Figure 11-3 Spectrum and I/Q Waveform (Span 25 MHz)



NOTE: A display with both an FFT derived spectrum in the upper window and an IQ Waveform in the lower window will appear when you activate a Complex Spectrum measurement. The active window is outlined in green. Changes to Frequency, Span or Amplitude will affect only the active window. Use the Next Window key to select a different window, and Zoom key to enlarge the window.

Step 8. Enable the IQ Waveform measurement:

Press **Meas**, **IQ Waveform**

Step 9. View the RF envelope:

Press **View/Display**, **RF Envelope**

Step 10. Set the analysis bandwidth:

Press **BW**, **Info BW**, 10, **MHz** (25 MHz if option B25 installed)

This view provides a waveform display of power versus time of the RF signal in the upper window with metrics for mean and peak-to-mean in the lower window. Refer to Figure 11-4 or Figure 11-5.

Figure 11-4 IQ Waveform Measurement - Time domain View (10 MHz BW)

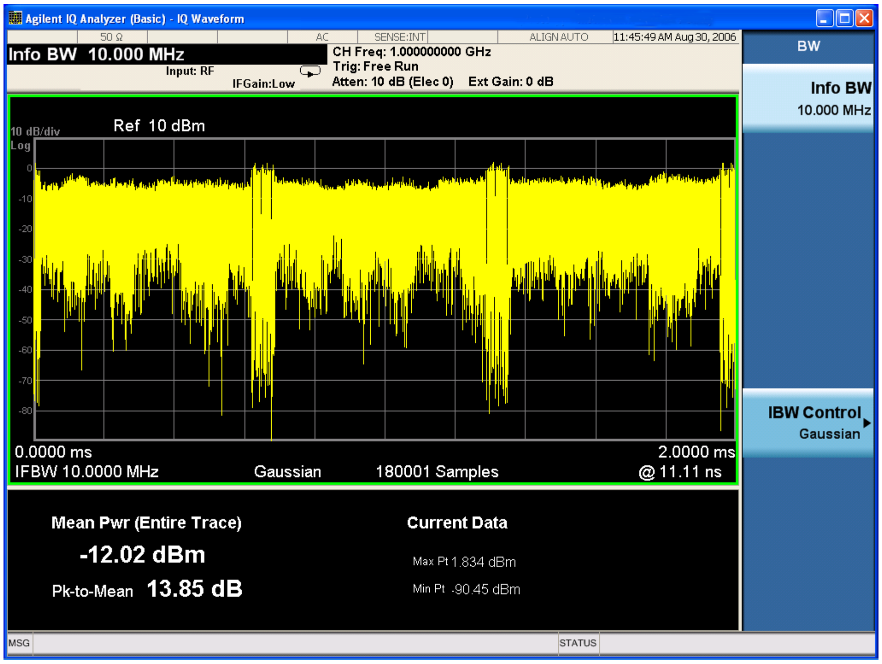
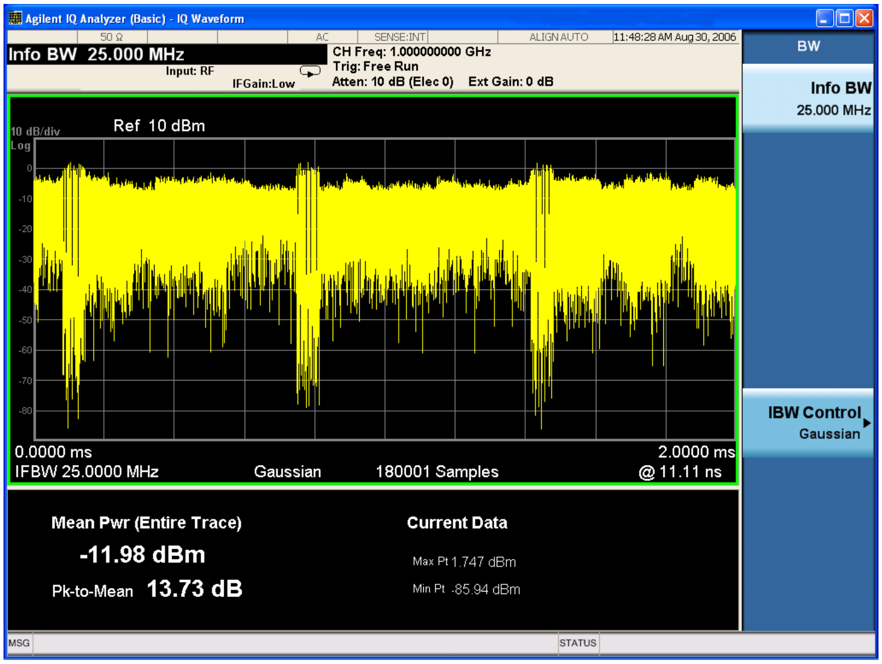


Figure 11-5 IQ Waveform Measurement - Time domain View (25 MHz BW)



Step 11. View the IQ Waveform:

Press **View/Display**, **IQ Waveform**

Step 12. Set the time scale:

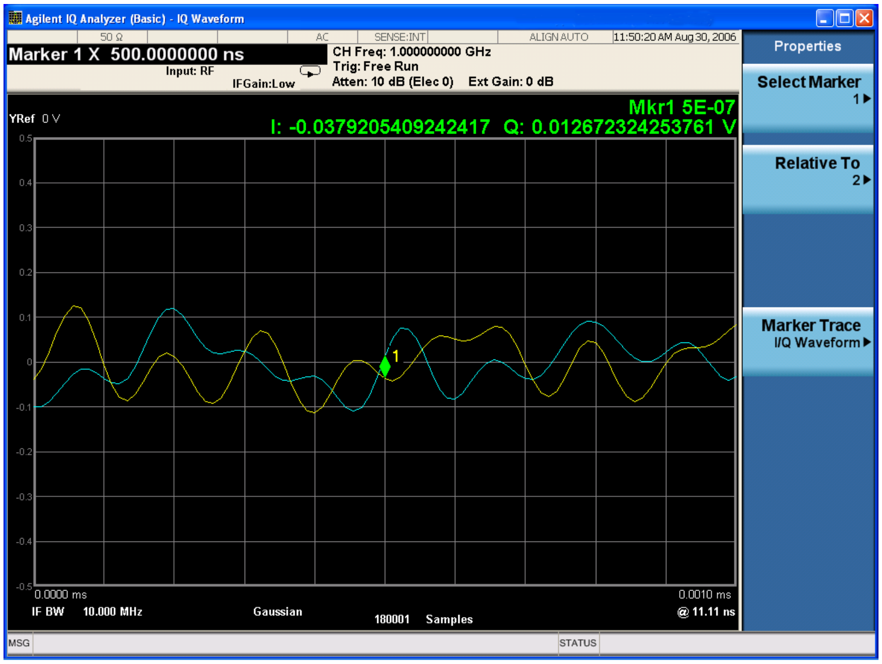
Press **Span X Scale**, **Scale/Div**, 100, **ns**

Step 13. Enable markers:

Press **Marker**, **Properties**, **Marker Trace**, **IQ Waveform**, 500, **ns**

This view provides a display of voltage versus time for the I and Q waveforms. Markers enable measurement of the individual values of I and Q. Refer to Figure 11-6.

Figure 11-6 IQ Waveform Measurement - with Markers



IQ Signal Analysis

12 Concepts

Resolving Closely Spaced Signals

Resolving Signals of Equal Amplitude

Two equal-amplitude input signals that are close in frequency can appear as a single signal trace on the analyzer display. Responding to a single-frequency signal, a swept-tuned analyzer traces out the shape of the selected internal IF (intermediate frequency) filter (typically referred to as the resolution bandwidth or RBW filter). As you change the filter bandwidth, you change the width of the displayed response. If a wide filter is used and two equal-amplitude input signals are close enough in frequency, then the two signals will appear as one signal. If a narrow enough filter is used, the two input signals can be discriminated and appear as separate peaks. Thus, signal resolution is determined by the IF filters inside the analyzer.

The bandwidth of the IF filter tells us how close together equal amplitude signals can be and still be distinguished from each other. The resolution bandwidth function selects an IF filter setting for a measurement. Typically, resolution bandwidth is defined as the 3 dB bandwidth of the filter. However, resolution bandwidth may also be defined as the 6 dB or impulse bandwidth of the filter.

Generally, to resolve two signals of equal amplitude, the resolution bandwidth must be less than or equal to the frequency separation of the two signals. If the bandwidth is equal to the separation and the video bandwidth is less than the resolution bandwidth, a dip of approximately 3 dB is seen between the peaks of the two equal signals, and it is clear that more than one signal is present.

For MXA Signal Analyzers in swept mode, sweep time is automatically set to a value that is inversely proportional to the square of the resolution bandwidth ($1/BW^2$),to keep the analyzer measurement calibrated. So, if the resolution bandwidth is reduced by a factor of 10, the sweep time is increased by a factor of 100 when sweep time and bandwidth settings are coupled. For the shortest measurement times, use the widest resolution bandwidth that still permits discrimination of all desired signals. Sweep time is also a function of which detector is in use, peak and normal detectors sweep as fast or more quickly than sample or average detectors. The MXA allows RBW selections up to 8 MHz in 1, 3, 10 steps and it has the flexibility to fine tune RBWs in increments of 10% for a total of 160 RBW settings.

For best sweep times and keeping the analyzer calibrated set the sweep time (**Sweep/Control**, **Sweep Time**) to **Auto**, and the sweep type (**Sweep/Control**, **Sweep Setup**, **Sweep Type**) to **Auto**. Use the widest resolution bandwidth and the narrowest span that still permits resolution of all desired signals.

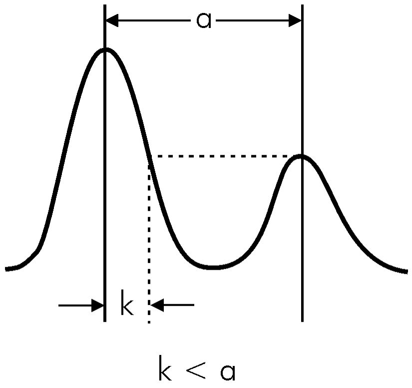
Resolving Small Signals Hidden by Large Signals

When dealing with the resolution of signals that are close together and not equal in amplitude, you must consider the shape of the IF filter of the analyzer, as well as its 3 dB bandwidth. (See "Resolving Signals of Equal Amplitude" on page 140 for more information.) The shape of a filter is defined by the selectivity, which is the ratio of the 60 dB bandwidth to the 3 dB bandwidth. If a small signal is too close to a larger signal, the smaller signal can be hidden by the skirt of the larger signal.

To view the smaller signal, select a resolution bandwidth such that k is less than a (see Figure 12-1). The separation between the two signals (a) must be greater than half the filter width of the larger signal (k), measured at the amplitude level of the smaller signal.

The digital filters in the MXA have filter widths about one-third as wide as typical analog RBW filters. This enables you to resolve close signals with a wider RBW (for a faster sweep time).

Figure 12-1 RBW Requirements for Resolving Small Signals



Trigger Concepts

NOTE The trigger functions let you select the trigger settings for a sweep or measurement. When using a trigger source other than Free Run, the analyzer will begin a sweep only with the selected trigger conditions are met. A trigger event is defined as the point at which your trigger source signal meets the specified trigger level and polarity requirements (if any). In FFT measurements, the trigger controls when the data is acquired for FFT conversion.

Selecting a Trigger

1. **Free Run Triggering**

   Pressing this key, when it is not selected, selects free-run triggering. Free run triggering occurs immediately after the sweep/measurement is initiated.

   Press **Trigger**, **Free Run**

2. **Video Triggering**

   The Video trigger condition is met when the video signal (the filtered and detected version of the input signal, including both RBW and VBW filtering) crosses the video trigger level. Video triggering triggers the measurement at the point at which the rising signal crosses the video trigger horizontal green line on the display:

   Press **Trigger**, **Video**, –**30**, **dBm**.

3. **External Triggering**

   In the event that you have an external trigger available that can be used to synchronize with the burst of interest, connect the trigger signal to the rear of the MXA using either the `Trigger 1 In` or `Trigger 2 In` input connector. It might be necessary to adjust the trigger level by rotating the front panel knob or by entering the numeric value on the keypad.

   Press **Trigger**, **External 1** or **External 2**, **Trigger Level**, and adjust as necessary.

4. **RF Burst Wideband Triggering**

   RF burst triggering occurs in the IF circuitry chain, as opposed to after the video detection circuitry with video triggering. In the event video triggering is used, the detection filters are limited to the maximum width of the resolution bandwidth filters. Set the analyzer in RF burst trigger mode.

   Press **Trigger**, **RF Burst**.

5. **Line Triggering**

   Line triggering selects the line signal as the trigger. A new sweep/measurement will start synchronized with the next cycle of the line voltage. Pressing this key, when it is already selected, access the line trigger setup menu.

   Press **Trigger**, **RF Burst**.

6. **Periodic Timer Triggering**

   This feature selects the internal periodic timer signal as the trigger. Trigger occurrences are set by the **Periodic Timer** parameter, which is modified by the **Sync Source** and **Offset**. Figure 12-2 shows the action of the periodic timer trigger. Before reviewing the figure, we'll explain some uses for the periodic trigger.

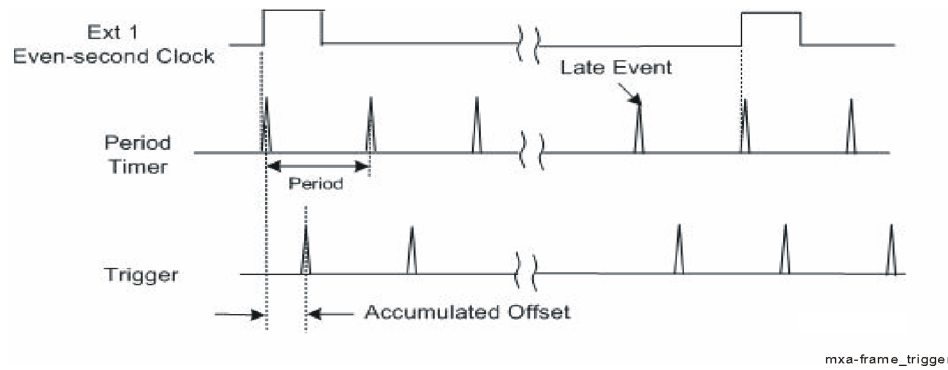
   A common application is measuring periodic burst RF signals for which a trigger signal is not easily available. For example, we might be measuring a TDMA radio which bursts every 20 ms. Let's assume that the 20 ms period is very consistent. Let's also assume that we do not have an external trigger source available that is synchronized with the period, and that the signal-to-noise ratio of the signal is not high enough to provide a clean RF burst trigger at all of the analysis frequencies. For example, we might want to measure spurious transmissions at an offset from the carrier that is larger than the bandwidth of the RF burst trigger. In this application, we can set the **Periodic Timer** to a 20.00 ms period and adjust the offset from that timer to position our trigger just where we want it. If we find that the 20.00 ms is not exactly right, we can adjust the period slightly to minimize the drift between the period timer and the signal to be measured.

   A second way to use this feature would be to use **Sync Source** temporarily, instead of **Offset**. In this case, we might tune to the signal in a narrow span and use the **RF Burst** trigger to synchronize the periodic timer. Then we would turn the sync source off so that it would not mistrigger. Mistriggering can occur when we are tuned so far away from the RF burst trigger that it is no longer reliable.

   A third example would be to synchronize to a signal that has a reference time element of much longer period than the period of interest. In some CDMA applications, it is useful to look at signals with a short periodicity, by synchronizing that periodicity to the "even-second clock" edge that happens every two seconds. Thus, we could connect the even-second clock trigger to Ext1 and use then Ext1 as the sync source for the periodic timer.

   The figure below illustrates this third example. The top trace represents the even-second clock. It causes the periodic timer to synchronize with the leading edge shown. The analyzer trigger occurs at a time delayed by the accumulated offset from the period trigger event. The periodic timer continues to run, and triggers continue to occur, with a periodicity determined by the analyzer time base. The timer output (labeled “late event”) will drift away from its ideal time due to imperfect matching between the time base of the signal being measured and the time base of the analyzer, and also because of imperfect setting of the period parameter. But the synchronization is restored on the next even-second clock event. (“Accumulated offset” is described in the in the **Offset** function section.)

Figure 12-2 Frame Triggering



a. **Period**

Sets the period of the internal periodic timer clock. For digital communications signals, this is usually set to the frame period of your current input signal. In the case that sync source is not set to OFF, and the external sync source rate is changed for some reason, the periodic timer is synchronized at the every external synchronization pulse by resetting the internal state of the timer circuit.

Press **Trigger**, **Periodic Timer**.

b. **Offset**

Adjusts the accumulated offset between the periodic timer events and the trigger event. Adjusting the accumulated offset is different than setting an offset, and requires explanation.

The periodic timer is usually unsynchronized with any external events, so the timing of its output events has no absolute meaning. Since the timing relative to external events (RF signals) is important, you need to be able to adjust (offset) it. However, you have no direct way to see when the periodic timer events occur. All that you can see is the trigger timing. When you want to adjust the trigger timing, you will be changing the internal offset between the periodic timer events and the trigger event. Because the absolute value of that internal offset is unknown, we will just call that the accumulated offset. Whenever the **Offset** parameter is changed, you are changing that accumulated offset. You can reset the displayed offset using **Reset Offset Display**. Changing the display does not change the value of the accumulated offset, and you can still make additional changes to accumulated offset.

Press **Trigger**, **Periodic Timer**.

c. **Reset Offset Display**

Resets the value of the periodic trigger offset display setting to 0.0 seconds. The current displayed trigger location may include an offset value defined with the **Offset** key. Pressing this key redefines the currently displayed trigger location as the new trigger point that is 0.0 s offset. The **Offset** key can then be used to add offset relative to this new timing

Press **Trigger**, **Periodic Timer**.

Concepts

AM and FM Demodulation Concepts

Demodulating an AM Signal Using the Analyzer as a Fixed Tuned Receiver (Time-Domain)

The zero span mode can be used to recover amplitude modulation on a carrier signal.

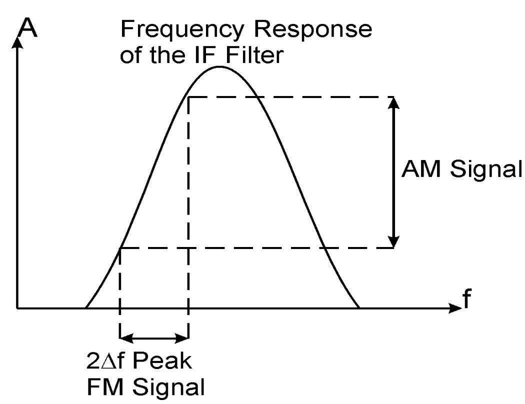
The following functions establish a clear display of the waveform:

- Triggering stabilizes the waveform trace by triggering on the modulation envelope. If the modulation of the signal is stable, video trigger synchronizes the sweep with the demodulated waveform.
- Linear display mode should be used in amplitude modulation (AM) measurements to avoid distortion caused by the logarithmic amplifier when demodulating signals.
- Sweep time to view the rate of the AM signal.
- RBW and VBW are selected according to the signal bandwidth.

Demodulating an FM Signal Using the Analyzer as a Fixed Tuned Receiver (Time-Domain)

To recover the frequency modulated signal, a spectrum analyzer can be used as a manually tuned receiver (zero span). However, in contrast to AM, the signal is not tuned into the passband center, but to one slope of the filter curve as Figure 12-3.

Figure 12-3 Determining FM Parameters using FM to AM Conversion



Here the frequency variations of the FM signal are converted into amplitude variations (FM to AM conversion) utilizing the slope of the selected bandwidth filter. The reason we want to measure the AM component is that the envelope detector responds only to AM variations. There are no changes in amplitude if the frequency changes of the FM signal are limited to the flat part of the RBW (IF filter). The resultant AM signal is then detected with the envelope detector and displayed in the time domain.

IQ Analysis Concepts

Purpose

IQ Analysis (Basic) mode is used to capture complex time domain data from wide bandwidth RF signals. This mode preserves the instantaneous vector relationships of time, frequency, phase and amplitude contained within the selected digitizer span or analysis BW, at the analyzer's center frequency, for output as IQ data. This IQ data can then be utilized internally or output over LAN, USB or GPIB for use with external analysis tools.

Within the IQ Analyzer mode, basic frequency domain, time domain and IQ measurements are available as initial signal and data verification tools in preparation for deriving the IQ data output. This is accomplished using the Complex Spectrum and IQ Waveform measurements. Although Complex Spectrum and IQ Waveform are defined as measurements in the IQ Analyses (Basic) mode, they act primarily as tools to verify the signals and data as stated above.

Complex Spectrum Measurement

Purpose

This measurement is FFT (Fast Fourier Transform) based. The FFT-specific parameters are located in the **Advanced** menu. The Complex Spectrum measurement provides a display in the upper window of power versus frequency with current (yellow trace) and average (blue trace) data. In addition, an IQ waveform of voltage versus time is provided in the lower window. One advantage of having an I/Q view available while in the spectrum measurement is that it allows you to view complex components of the same signal without changing settings or measurements.

Measurement Method

The measurement uses digital signal processing to sample the input signal and convert it to the frequency domain. With the instrument tuned to a fixed center frequency, samples are digitized at a high rate, converted to I and Q components with DSP hardware, and then converted to the frequency domain with FFT software.

Troubleshooting Hints

Changes made by the user to advanced spectrum settings, particularly to ADC range settings, can inadvertently result in spectrum measurements that are invalid and cause error messages to appear. Care needs to be taken when using advanced features.

IQ Waveform Measurement

Purpose

The IQ Waveform measurement provides a time domain view of the RF signal envelope with power versus time or an IQ waveform with the I and Q signal waveforms in parameters of voltage versus time. The RF Envelope view provides the power verses time display, and the I/Q Waveform view provides the voltage versus time display. One advantage of having an I/Q Waveform view available while making a waveform measurement is that it allows you to view complex components of the same signal without changing settings or measurements.

The waveform measurement can be used to perform general purpose power measurements in the time domain with excellent accuracy.

Measurement Method

The instrument makes repeated power measurements at a set frequency, similar to the way a swept-tuned signal analyzer makes zero span measurements. The input analog signal is converted to a digital signal, which then is processed into a representation of a waveform measurement. The measurement relies on a high rate of sampling to create an accurate representation of a time domain signal.

Spurious Emissions Measurement Concepts

Purpose

Spurious signals can be caused by different combinations of signals in the transmitter. The spurious emissions from the transmitter should be minimized to guarantee minimum interference with other frequency channels in the system. Harmonics are distortion products caused by nonlinear behavior in the transmitter. They are integer multiples of the transmitted signal carrier frequency.

This measurement verifies the frequency ranges of interest are free of interference by measuring the spurious signals specified by the user defined range table.

Measurement Method

The table-driven measurement has the flexibility to set up custom parameters such as frequency, span, resolution bandwidth, and video bandwidth. Up to the top 40 spurs can be viewed

For each range that you specify and activate, the analyzer scans the band using the specified Range Table settings. Then using the Peak Excursion and Peak Threshold values determines which spurs to report.

As each band is swept, any signal which is above the Peak Threshold value and has a peak excursion of greater than the Peak Excursion value will be added to a list of spurs displayed in the lower results window. A total of 200 spurs can be recorded for one measurement, with a limit of 10 spurs per frequency range. To improve repeatability, you can increase the number of averages.

From the spurs in the list, those with peak amplitude greater than the Absolute Limit for that range will be logged as a measurement failure and denoted by an 'F' in the 'Amplitude' column of the table. If no spurs are reported, but the measured trace exceeds the limit line for any range, the fail flag is set to fail.

This measurement has the ability to display two traces using different detectors on the display simultaneously. All spur detection and limit line testing are only applied to the trace associated with Detector 1, which will be colored yellow. The trace associated with Detector 2 will be colored cyan.

If the sweep time for the range exceeds 2 seconds, a flashing message “Sweeping...Please Wait” will appear in the annunciator area. This advises you that the time to complete the sweep is between 2 and 2000 seconds, and is used as without it the display would appear stagnant and you may think the measurement is not functional.

Concepts

Spectrum Emission Mask Measurement Concepts

Purpose

The Spectrum Emission Mask measurement includes the in-band and out-of-band spurious emissions. As it applies to W-CDMA (3GPP), this is the power contained in a specified frequency bandwidth at certain offsets relative to the total carrier power. It may also be expressed as a ratio of power spectral densities between the carrier and the specified offset frequency band.

This spectrum emission mask measurement is a composite measurement of out-of-channel emissions, combining both in-band and out-of-band specifications. It provides useful figures-of-merit for the spectral regrowth and emissions produced by components and circuit blocks, without the rigor of performing a full spectrum emissions mask measurement.

Measurement Method

The spectrum emission mask measurement measures spurious signal levels in up to five pairs of offset/region frequencies and relates them to the carrier power. The reference channel integration bandwidth method is used to measure the carrier channel power and offset/region powers. When **Offset** is selected, spectrum emission mask measurements are made, relative to the carrier channel frequency bandwidth. When **Region** is selected, spurious emission absolute measurements are made, set by specifying start and stop RF frequencies. The upper frequency range limit is 3.678 GHz.

This integration bandwidth method is used to perform a data acquisition. In this process, the reference channel integration bandwidth (Meas BW) is analyzed using the automatically defined resolution bandwidth (Res BW), which is much narrower than the channel bandwidth. The measurement computes an average power of the channel or offset/region over a specified number of data acquisitions, automatically compensating for resolution bandwidth and noise bandwidth.

This measurement requires the user to specify the measurement bandwidths of carrier channel and each of the offset/region frequency pairs up to 5. Each pair may be defined with unique measurement bandwidths. The results are displayed both as relative power in dBc, and as absolute power in dBm.

Occupied Bandwidth Measurement Concepts

Purpose

Occupied bandwidth measures the bandwidth containing 99.0 of the total transmission power.

The spectrum shape of a signal can give useful qualitative insight into transmitter operation. Any distortion to the spectrum shape can indicate problems in transmitter performance.

Measurement Method

The instrument uses digital signal processing (DSP) to sample the input signal and convert it to the frequency domain. With the instrument tuned to a fixed center frequency, samples are digitized at a high rate with DSP hardware, and then converted to the frequency domain with FFT software.

The total absolute power within the measurement frequency span is integrated for its 100% of power. The lower and upper frequencies containing 0.5% each of the total power are then calculated to get 99.0% bandwidth.

13 Programming Examples

Programming Examples Information and Requirements

- The programming examples were written for use on an IBM compatible PC.
- The programming examples use C, Visual Basic, or VEE programming languages.
- The programming examples use VISA interfaces (GPIB, LAN, or USB).
- Some of the examples use the IVI-COM drivers.

  Interchangeable Virtual Instruments COM (IVI-COM) drivers: Develop system automation software easily and quickly. IVI-COM drivers take full advantage of application development environments such as Visual Studio using Visual Basic, C# or Visual C++ as well as Agilent's Test and Measurement Toolkit. You can now develop application programs that are portable across computer platforms and I/O interfaces. With IVI-COM drivers you do not need to have in depth test instrument knowledge to develop sophisticated measurement software. IVI-COM drivers provide a compatible interface to all. COM environments. The IVI-COM software drivers can be found at the URL: http://www.agilent.com/find/ivi-com
- Most of the examples are written in C, Visual Basic, VEE, or LabVIew using the Agilent VISA transition library.

  The Agilent I/O Libraries Suite must be installed and the GPIB card, USB to GPIB interface, or Lan interface USB interface configured. The latest Agilent I/O Libraries Suite is available: www.agilent.com/find/iolib
- The STATus subsystem of commands is used to monitor and query hardware status. These hardware registers monitor various events and conditions in the instrument. Details about the use of these commands and registers can be found in the manual/help in the Utility Functions section on the STATus subsystem.

Programming Examples

Available Programing Examples

The following examples work with a MXA Signal Analyzer. These examples use one of the following programming languages: Visual Basic® 6, Visual Basic.NET®, MS Excel®, C++, ANSI C, C#.NET, and Agilent VEE Pro.

These examples are available in either the "progexamples" directory on the Agilent Technologies MXA Signal Analyzer documentation CD-ROM or the "progexamples" directory in the analyzer. The file names for each example is listed at the end of the example description. The examples can also be found on the Agilent Technologies, Inc. web site at URL:

http://www.agilent.com/find/sa_programming

NOTE These examples have all been test and validated as functional in the Spectrum Analyzer mode. They have not been tested in all other modes. However, they should work in all other modes except where exceptions are noted.

Programming using Visual Basic® 6, Visual Basic.NET® and MS Excel®:

- *Transfer Screen Images* from your MXA Signal Analyzer using Visual Basic 6

  This example program stores the current screen image on the instrument flash memory as "D:\PICTURE.PNG". It then transfers the image over GPIB or LAN and stores the image on your PC in the current directory as "PICTURE.PNG". The file "D:\PICTURE.PNG" is then deleted on the instrument flash memory.

  File name: MXA_screen.bas

- *Binary Block Trace* data transfer from your MXA Signal Analyzer using Visual Basic 6

  This example program queries the IDN string from the instrument and then reads the trace data in Spectrum Analysis mode in binary format (Real,32 or Real,64 or Int,32). The data is then stored to a file "bintrace.txt". This data transfer method is faster than the default ASCII transfer mode, because less data is sent over the bus.

  File name: bintrace.bas

Programming using C++, ANSI C and C#.NET*:*

- *Serial Poll for Sweep Complete* using C++

  This example demonstrates how to:

1. Perform an instrument sweep.
2. Poll the instrument to determine when the operation is complete.
3. Perform an instrument sweep.

File name: MXA_Sweep.c

- *Service Request Method (SRQ)* determines when a measurement is done by waiting for SRQ and reading Status Register using C++.

  This example demonstrates how:

  1. Set the service request mask to assert SRQ when either a measurement is uncalibrated or an error message has occurred,
  2. Initiate a sweep and wait for the SRQ interrupt,
  3. Poll all instruments and report the nature of the * interrupt on the signal analyzer.

  The STATus subsystem of commands is used to monitor and query hardware status. These hardware registers monitor various events and conditions in the instrument. Details about the use of these commands and registers can be found in the manual/help in the Utility Functions section on the STATus subsystem.

  File name: MXA_SRQ.C

- *Relative Band Power Markers* using C++

  This example demonstrates how to set markers as Band Power Markers and obtain their band power relative to another specified marker.

  File name: MXA_BPM.c

- *Trace Detector / Couple Markers* using C++

  This example demonstrates how to:

  1. Set different types of traces (max hold, clear and write, min hold)
  2. Set markers to specified traces
  3. Couple markers

  Note: The MXA Signal Analyzer is capable of multiple simultaneous detectors (i.e. peak detector for max hold, sample for clear and write, and negative peak for min hold).

  File name: MXA_tracecouple.c

- *Phase Noise* using C++

  This example demonstrates how to:

  1. Remove instrument noise from the phase noise
  2. Calculate the power difference between 2 traces

  File name: MXA_phasenoise.c

Programming using Agilent VEE Pro*:*

- *Transfer Screen Images* from my MXA Signal Analyzer using Agilent VEE Pro

  This example program stores the current screen image on the instrument flash memory as “`D:\mxascr.png`”. It then transfers the image over GPIB and stores the image on your PC in the desired directory as “`capture.gif`”. The file “`D:\mxascr.png`” is then deleted on the instrument flash memory.

  File name: MXA_ScreenCapture.vee

- *Transfer Trace Data* data transfer using Agilent VEE Pro

  This example program transfers the trace data from your MXA Signal Analyzer. The program queries the IDN string from the instrument and supports Integer 32, real 32, real 64 and ASCII data. The program returns 1001 trace points for the MXA signal analyzer.

  File name: transfertrace.vee

Programming Fundamentals

SCPI Language Basics

This section is not intended to teach you everything about the SCPI (Standard Commands for Programmable Instruments) programming language. The SCPI Consortium or IEEE can provide that level of detailed information. For more information refer to the websites for the IEEE Standard 488.1 (IEEE Standard Digital Interface for Programmable Instrumentation).

Topics covered in this chapter include:

- "Command Keywords and Syntax" on page 161
- "Creating Valid Commands" on page 161
- "Special Characters in Commands" on page 162
- "Parameters in Commands" on page 163
- "Putting Multiple Commands on the Same Line" on page 166

Command Keywords and Syntax

A typical command is made up of keywords set off by colons. The keywords are followed by parameters that can be followed by optional units.

Example: `SENSe:FREQuency:STARt 1.5 MHZ`

The instrument does not distinguish between upper and lower case letters. In the documentation, upper case letters indicate the short form of the keyword. The lower case letters, indicate the long form of the keyword. Either form may be used in the command.

Example: `Sens:Freq:Star 1.5 mhz`

is the same as `SENSE:FREQ:start 1.5 MHz`

NOTE The command `SENS:FREQU:STAR` would not be valid because `FREQU` is neither the short, nor the long form of the command. Only the short and long forms of the keywords are allowed in valid commands.

Creating Valid Commands

Commands are not case sensitive and there are often many different ways of writing a particular command. These are examples of valid

Programming Examples

commands for a given command syntax:

<table>
<tr><th>Command Syntax</th><th>Sample Valid Commands</th></tr>
<tr><td>[SENSe:]BANDwidth[:RESolution] <freq></td><td>The following sample commands are all identical. They will all cause the same result.
• Sense:Band:Res 1700
• BANDWIDTH:RESOLUTION 1.7e3
• sens:band 1.7KHZ
• SENS:band 1.7E3Hz
• band 1.7kHz
• bandwidth:RES 1.7e3Hz</td></tr>
<tr><td>MEASure:SPECtrum[n]?</td><td>• MEAS:SPEC?
• Meas:spec?
• meas:spec3?
The number 3 in the last meas example causes it to return different results then the commands above it. See the command description for more information.</td></tr>
<tr><td>[:SENSe]:DETector[:FUNCtion] NEGative|POSitive|SAMPle</td><td>• DET:FUNC neg
• Detector:Func Pos</td></tr>
<tr><td>INITiate:CONTinuous ON|OFF|1|0</td><td>The sample commands below are identical.
• INIT:CONT ON
• init:continuous 1</td></tr>
</table>

Special Characters in Commands

<table>
<tr><th>Special Character</th><th>Meaning</th><th>Example</th></tr>
<tr><td>|</td><td>A vertical stroke between parameters indicates alternative choices. The effect of the command is different depending on which parameter is selected.</td><td>Command: TRIGger:SOURce EXTernal|INTernal|LINE
The choices are external, internal, and line.
Ex: TRIG:SOURCE INT
is one possible command choice.</td></tr>
<tr><td></td><td>A vertical stroke between keywords indicates identical effects exist for both keywords. The command functions the same for either keyword. Only one of these keywords is used at a time.</td><td>Command: SENSe:BANDwidth|BWIDth:OFFSet
Two identical commands are:
Ex1: SENSE:BWIDTH:OFFSET
Ex2: SENSE:BAND:OFFSET</td></tr>
</table>

Special Character	Meaning	Example
[ ]	Keywords in square brackets are optional when composing the command. These implied keywords will be executed even if they are omitted.	Command: [SENSe:]BANDwidth[:RESolution]:AUTO The following commands are all valid and have identical effects: Ex1: bandwidth:auto Ex2: band:resolution:auto Ex3: sense:bandwidth:auto
< >	Angle brackets around a word, or words, indicates they are not to be used literally in the command. They represent the needed item.	Command: SENS:FREQ <freq> In this command example the word <freq> should be replaced by an actual frequency. Ex: SENS:FREQ 9.7MHz.
{ }	Parameters in braces can optionally be used in the command either not at all, once, or several times.	Command: MEASure:BW <freq>{,level} A valid command is: meas:BW 6 MHz, 3dB, 60dB

Parameters in Commands

There are four basic types of parameters: booleans, keywords, variables and arbitrary block program data.

OFF|ON|0|1 (Boolean)
This is a two state boolean-type parameter. The numeric value 0 is equivalent to OFF. Any numeric value other than 0 is equivalent to ON. The numeric values of 0 or 1 are commonly used in the command instead of OFF or ON. Queries of the parameter always return a numeric value of 0 or 1.

keyword
The keywords that are allowed for a particular command are defined in the command syntax description.

Units
Numeric variables may include units. The valid units for a command depend on the variable type being used. See the following variable descriptions. The indicated default units will be used if no units are sent. Units can follow the numerical value with, or without, a space.

Variable
A variable can be entered in exponential format as well as standard numeric format. The appropriate range of the variable and its optional units are defined in the command description.

The following keywords may also be used in commands, but not all commands allow keyword variables.

- DEFault - resets the parameter to its default value.
- UP - increments the parameter.
- DOWN - decrements the parameter.
- MINimum - sets the parameter to the smallest possible value.
- MAXimum - sets the parameter to the largest possible value.

The numeric value for the function's MINimum, MAXimum, or DEFault can be queried by adding the keyword to the command in its query form. The keyword must be entered following the question mark.

Example query: `SENSE:FREQ:CENTER? MAX`

Variable Parameters

<integer>	is an integer value with no units.
<real>	Is a floating point number with no units.
<freq> <bandwidth>	Is a positive rational number followed by optional units. The default unit is Hertz. Acceptable units include: Hz, kHz, MHz, GHz.
<time> <seconds>	Is a rational number followed by optional units. The default units are seconds. Acceptable units include: ks, s, ms, µs, ns.
<voltage>	Is a rational number followed by optional units. The default units are Volts. Acceptable units include: V, mV, µV, nV
<current>	Is a rational number followed by optional units. The default units are Amperes. Acceptable units include: A, mA, µA, nA.
<power>	Is a rational number followed by optional units. The default units are W. Acceptable units include: mAW, kW, W, mW, µW, nW, pW.
<ampl>	Is a rational number followed by optional units. The default units are dBm. Acceptable units include: dBm, dBmV, dBµV.
<rel_power> <rel_ampl>	Is a positive rational number followed by optional units. The default units are dB. Acceptable units include: dB.
<percent>	Is a rational number between 0 and 100. You can either use no units or use PCT.

<angle>
<degrees> Is a rational number followed by optional units. The default units are degrees. Acceptable units include: DEG, RAD.

<string> Is a series of alpha numeric characters.

<bit_pattern> Specifies a series of bits rather than a numeric value. The bit series is the binary representation of a numeric value. There are no units.

Bit patterns are most often specified as hexadecimal numbers, though octal, binary or decimal numbers may also be used. In the SCPI language these numbers are specified as:

- Hexadecimal, #Hdddd or #hdddd where ‘d’ represents a hexadecimal digit 0 to 9 and ‘a’ to ‘f’. So #h14 can be used instead of the decimal number 20.
- Octal, #Odddddd or #odddddd where ‘d’ represents an octal digit 0 to 7. So #o24 can be used instead of the decimal number 20.
- Binary, #Bdddddddddddddddd or #bdddddddddddddddd where ‘d’ represents a 1 or 0. So #b10100 can be used instead of the decimal number 20.

Block Program Data

Some parameters consist of a block of data. There are a few standard types of block data. Arbitrary blocks of program data can also be used.

<trace> Is an array of rational numbers corresponding to displayed trace data. See FORMat:DATA for information about available data formats.

A SCPI command often refers to a block of current trace data with a variable name such as: Trace1, Trace2, or Trace3, depending on which trace is being accessed.

<arbitrary block data> Consists of a block of data bytes. The first information sent in the block is an ASCII header beginning with #. The block is terminated with a semi-colon. The header can be used to determine how many bytes are in the data block. There are no units. You will not get block data if your data type is ASCII, using `FORMat:DATA ASCII` command. Your data will be comma separated ASCII values.

Block data example: suppose the header is #512320.

- The first digit in the header (5) tells you how many additional digits/bytes there are in the header.

- The 12320 means 12 thousand, 3 hundred, 20 data bytes follow the header.
- Divide this number of bytes by your current data format (bytes/data point), either 8 (for real,64), or 4 (for real,32). For this example, if you are using real64 then there are 1540 points in the block.

Putting Multiple Commands on the Same Line

Multiple commands can be written on the same line, reducing your code space requirement. To do this:

- Commands must be separated with a semicolon (;).
- If the commands are in different subsystems, the key word for the new subsystem must be preceded by a colon (:).
- If the commands are in the same subsystem, the full hierarchy of the command key words need not be included. The second command can start at the same key word level as the command that was just executed.

SCPI Termination and Separator Syntax

All binary trace and response data is terminated with <NL><END>, as defined in Section 8.5 of IEEE Standard 488.2-1992, *IEEE Standard Codes, Formats, Protocols and Common Commands for Use with ANSI/IEEE Std 488.1-1987*. New York, NY, 1992. (Although one intent of SCPI is to be interface independent, <END> is only defined for IEEE 488 operation.)

The following are some examples of good and bad commands. The examples are created from a theoretical instrument with the simple set of commands indicated below:

```
[:SENSe]
     :POWer
          [:RF]
          :ATTenuation 40dB

:TRIGger
     [:SEQuence]
     :EXTernal [1]
          :SLOPe
               POSitive

[:SENSe]
     :FREQuency
          :STARt
     :POWer
     [:RF]
          :MIXer
               :RANGe
               [:UPPer]
```


Programming Examples

Bad Command	Good Command
PWR:ATT 40dB	POW:ATT 40dB
The short form of POWER is POW, not PWR.	
FREQ:STAR 30MHz;MIX:RANG -20dBm	FREQ:STAR 30MHz;POW:MIX:RANG -20dBm
The MIX:RANG command is in the same :SENSE subsystem as FREQ, but executing the FREQ command puts you back at the SENSE level. You must specify POW to get to the MIX:RANG command.	
FREQ:STAR 30MHz;POW:MIX RANG -20dBm	FREQ:STAR 30MHz;POW:MIX:RANG -20dBm
MIX and RANG require a colon to separate them.	
:POW:ATT 40dB;TRIG:FREQ:STAR 2.3GHz	:POW:ATT 40dB;:FREQ:STAR 2.3GHz
:FREQ:STAR is in the :SENSE subsystem, not the :TRIGGER subsystem.	
:POW:ATT?:FREQ:STAR?	:POW:ATT?;:FREQ:STAR?
:POW and FREQ are within the same :SENSE subsystem, but they are two separate commands, so they should be separated with a semicolon, not a colon.	
:POW:ATT -5dB;:FREQ:STAR 10MHz	:POW:ATT 5dB;:FREQ:STAR 10MHz
Attenuation cannot be a negative value.	

Improving Measurement Speed

There are a number of things you can do in your programs to make them run faster:

"Turn off the display updates" on page 168

"Use binary data format instead of ASCII" on page 168

"Minimize the number of GPIB transactions" on page 169

"Consider using USB or LAN instead of GPIB" on page 170

"Minimize DUT/instrument setup changes" on page 170

"Avoid automatic attenuator setting" on page 170

"Put ADC Ranging in Bypass for FFT Measurements" on page 171

"Avoid using RFBurst trigger for single burst signals" on page 171

"N9071A:Optimize your GSM output RF spectrum switching measurement" on page 172

"Making power measurements on multiple bursts or slots? Use CALCulate:DATA<n>:COMPress?" on page 172

Turn off the display updates

`:DISPlay:ENABle OFF` turns off the display. That is, the data may still be visible, but it will no longer be updated. Updating the display slows down the measurement. For remote testing, since the computer is processing the data rather than a person, there is no need to display the data on the analyzer screen.

Use binary data format instead of ASCII

The ASCII data format is the instrument default since it is easier for people to understand and is required by SCPI for *RST. However, data input/output is faster using the binary formats.

`:FORMat:DATA REAL,64` selects the 64-bit binary data format for all your numerical data queries. You may need to swap the byte order if you are using a PC rather than UNIX. **`NORMal`** is the default byte order. Use **`:FORMat:BORDer SWAP`** to change the byte order so that the least significant byte is sent first. (Real,32 which is smaller and somewhat faster, should only be used if you do not need full resolution for your data. Some frequency data may require full 64-bit resolution.)

When using the binary format, data is sent in a block of bytes with an ASCII header. A data query would return the block of data in the following format: `#DNNN<nnn binary data bytes>`

To parse the data:

- Read two characters (#D), where D tells you how many N characters follow the D character.

- Read D characters, the resulting integer specifies the number of data bytes sent.
- Read the bytes into a real array.

For example, suppose the header is #512320.

- The first character/digit in the header (5) tells you how many additional digits there are in the header.
- The 12320 means 12 thousand, 3 hundred, 20 data bytes follow the header.
- Divide this number of bytes by your current data format (bytes/data point), 8 for real,64. For this example, there are 1540 data points in the block of data.

Minimize the number of GPIB transactions

When you are using the GPIB for control of your instrument, each transaction requires driver overhead and bus handshaking, so minimizing these transactions reduces the time used.

- You can reduce bus transactions by sending multiple commands per transaction. See the information on “Putting Multiple Commands on the Same Line” in the SCPI Language Basics section.

- If you are making the same measurement multiple times with small changes in the measurement setup, use the READ command. It is faster then using INITiate and FETCh.

- If you are changing the frequency and making a measurement repeatedly, you can reduce transactions by sending the optional frequency parameter with your READ command.

  (for example, READ:<meas>? {<freq>} These optional parameters are not available in some modes such as Spectrum Analysis or Phase Noise.

  The CONFigure/MEASure/READ commands for measurements in the option Modes allow you to send center frequency setup information along with the command. (for example, `MEAS:PVT? 935.2MHz`) This sets the power vs. time measurement to its defaults, then changes the center frequency to 935.2 MHz, initiates a measurement, waits until it is complete and returns the measurement data.

- If you are doing bottoms/middle/top measurements on base stations, you can reduce transactions by making a time slot active at each of the B,M,T frequencies. Then issue three measurement at once in the programming code and retrieve three data sets with just one GPIB transaction pair (write, read).

  For example, send READ:PFER? <Freq_bottom>>;PFER? <Freq_middle>;PFER? <Freq_top> This single transaction initiates three different phase and frequency error measurements at each of the three different frequencies provided and returns the data. Then you can read the three sets of data.

Consider using USB or LAN instead of GPIB

USB and LAN allow faster data input and output. This is especially important if you are moving large blocks of data. You will not get this improved throughput using LAN if there is excessive LAN traffic (that is, your test instrument is connected to a very busy enterprise LAN). You may want to use a private LAN that is only for your test system.

Minimize DUT/instrument setup changes

- Some instrument setup parameters are common to multiple measurements. You should look at your measurement process with an eye toward minimizing setup changes. If your test process involves nested loops, make sure that the inner-most loop is the fastest. Also, check if the loops could be nested in a different order to reduce the number of parameter changes as you step through the test.
- Are you are using the measurements under the **Meas** key? Remember that if you have already set your Meas Setup parameters for a measurement, and you want to make another one of these measurements later, use READ:<meas>?. The MEASure:<meas>?. command resets all the settings to the defaults, while READ changes back to that measurement without changing the setup parameters from the previous use.
- Are you are using the Measurements under the **Meas** key? Remember that *Mode Setup* parameters remain constant across all the measurements in that mode (for example, center/channel frequency, amplitude, radio standard, input selection, trigger setup). You do not have to re-initialize them each time you change to a different measurement.

Avoid unnecessary use of *RST

Remember that while *RST does not change the current Mode, it presets all the measurements and settings to their factory defaults. This forces you to reset your analyzer's measurement settings even if they use similar mode setup or measurement settings. See Minimize DUT/instrument setup changes. (Also note that *RST may put the instrument in single measurement/sweep for some modes.)

Avoid automatic attenuator setting

Many of the one-button measurements use an internal process for automatically setting the value of the attenuator. It requires measuring an initial burst to identify the proper attenuator setting before the next burst can be measured properly. If you know the amount of attenuation or the signal level needed for your measurement, just set it.

Note that spurious types of measurements must be done with the attenuator set to automatic (for measurements like: output RF spectrum, transmit spurs, adjacent channel power, spectrum emission mask). These types of measurements start by tuning to the signal, then they tune away from it and must be able to reset the attenuation value as needed.

Put ADC Ranging in Bypass for FFT Measurements

Setting ADC ranging to the Buypass mode can speed FFT measurements up by 10% to 50%. (Use ADC:RANG NONE) Bypass allows triggered FFT measurement to occur at the trigger time instead of following an autoranging time, so it can improve measurement speed. It does, however, add additonal noise degrading your signal to noise level, so it should be used carefully.

Avoid using RFBurst trigger for single burst signals

RFBurst triggering works best when measuring signals with repetitive bursts. For a non-repetitive or single burst signals, use the IF (video) trigger or external trigger, depending on what you have available.

RFBurst triggering depends on its establishment of a valid triggering reference level, based on previous bursts. If you only have a single burst, the peak detection nature of this triggering function, may result in the trigger being done at the wrong level/point generating incorrect data, or it may not trigger at all.

Are you making a single burst measurement? To get consistent triggering and good data for this type of measurement application, you need to synchronize the triggering of the DUT with the analyzer. You should use the analyzer’s internal status system for this.

The first step in this process is to initialize the status register mask to look for the “waiting for trigger” condition (bit 5). Use **`:STATus:OPERation:ENABle 32`**

Then, in the measurement loop:

1. **`:STATus:OPERation:EVENt?`** This query of the operation event register is to clear the current register contents.

2. **`:READ:PVT?`** initiates a measurement (in this example, for GSM power versus time) using the previous setup. The measurement will then be waiting for the trigger.

   Make sure the attenuation is set manually. Do NOT use automatic attenuation as this requires an additional burst to determine the proper attenuation level before the measurement can be made.

3. Create a small loop that will serial poll the instrument for a status byte value of binary 128. Then wait 1 msec (100 ms if the display is

left on/enabled) before checking again, to keep the bus traffic down. These two commands are repeated until the condition is set, so we know that the trigger is armed and ready.

4. Trigger your DUT to send the burst.
5. Return the measurement data to your computer.

NOTE This process cannot be done with the current VXI plug-n-play driver implementation. You will need to use the previous SCPI commands.

N9071A:Optimize your GSM output RF spectrum switching measurement

For ORFS (switching), setting the break frequency to zero (0) puts the analyzer in a measurement setup where it can use a direct time measurement algorithm, instead of an FFT-based algorithm. This non-FFT approach is faster. (However, remember that your break frequency for ORFS (modulation) measurements must be >400 kHz for valid measurements, so you will need to change the break frequency if you are making both types of measurements.)

Making power measurements on multiple bursts or slots? Use CALCulate:DATA<n>:COMPress?

The CALC:DATA:COMP? query is the fastest way to measure power data for multiple bursts/slots. There are two reasons for this:

1. it can be used to measure data across multiple, consecutive slots/frames with just one measurement, instead of a separate measurement on each slot
2. it can pre-process and/or decimate the data so that you only return the information that you need which minimizes data transfer to the computer

For example: if you want to do a power measurement for a GSM base station where you generate a repeating frame with 8 different power levels, you can gather all the data with a single **`CALC:DATA:COMP?`** acquisition, using the waveform measurement.

With **`CALC:DATA2:COMP? MEAN,9,197,1730`** you can measure the mean power in those bursts. This single command will measure the data across all 8 frames, locate the first slot/burst in each of the frames, calculate the mean power of those bursts, then return the resulting 8 values.

NOTE For later version of firmware (after A.02.00) you can use equivalent time values for the CALC:DATA<n>:COMP? query. The command would then be **`CALC:DATA2:COMP? MEAN,25us,526us,579.6us,8`**

Use the following to set up the GSM Waveform measurement:

- **:CONF:WAV?** turns on the waveform measurement
- **:WAV:BAND 300khz** sets a resolution bandwidth of 300 kHz
- **:WAV:SWE:TIME 5ms** sets a sweep time of 5 milliseconds
- **:WAV:BAND:TYPE FLAT** selects the flat filter type
- **:WAV:DEC 4;DEC:STAT ON** selects a decimation of 4 and turns on decimation. This reduces the amount of data that needs to be sent since the instrument hardware decimates (throws some away).
- **:INIT** to initiate a measurement and acquire the data
- **CALC:DATA2:COMP? MEAN,25us,526us,579.6us,8** to return the desired data

There are two versions of this command depending on your firmware revision. Earlier revisions require the optional variables be entered in terms of their position in the trace data array. Current instruments allow the variables to be entered in terms of time.

For early firmware revisions you need to know the sample interval. In the waveform measurement it is equal to the aperture value. Query :WAVeform:APERture? to find the sample interval. (Note: the WAV:APER? command always takes decimation into account.) The sample interval (aperture value) is dependent on the settings for resolution bandwidth, filter type, and decimation. See Table • to see how these values relate.

The parameters for this GSM example are:
MEAN,9,197,1730 (or with later firmware:
MEAN,25us,526us,579.6us,8)

- MEAN calculates the mean of the measurement points indicated
- 9 is how many points you want to discard before you look at the data. This allows you to skip over any “unsettled” values at the beginning of the burst. You can calculate this start offset by (25µs/sampleInterval)l
- 197 is the length of the data you want to use. This would be the portion of the burst that you want to find the mean power over. You can calculate this length by (526µs/sampleInterval)
- 1730 is how much data you have before you repeat the process. For this example it is the time between the start offset point on the burst in the first slot (first frame) to the same spot on the burst in the first slot (second frame). You can calculate this by (576.9µs*N/sampleInterval) where N is the number of data items that you want. In this case it is the number of slots in the frame, N=8.)

Table 13-1 GSM Parameters for 1 Slot/Frame Measurement Requirements

Resolution Bandwidth	Filter Type	Decimation	Aperture	Start	Length	Repeat
500 or 300 kHz	Flat or Gaussian	4 or 1	dependent on settings	24 µsec[a]	526 µsec[a]	576.9 µsec[a]

Programming Examples

Table 13-1 GSM Parameters for 1 Slot/Frame Measurement Requirements

Resolution Bandwidth	Filter Type	Decimation	Aperture	Start	Length	Repeat
500 kHz	Gaussian	1	0.2 µsec	124	2630	2884.6
500 kHz	Gaussian	4	0.8 µsec	31	657	721.15
500 kHz	Flat	1	0.4 µsec	61	1315	1442.3
500 kHz	Flat	4	1.6 µsec	15	329	360.575
300 kHz	Gaussian	1	0.2667 µsec	90	1972	2163.1
300 kHz	Gaussian	4	1.07 µsec	22	492	539.16
300 kHz	Flat	1	0.6667 µsec	36	789	865.31
300 kHz	Flat	4	2.667 µsec	9	197	216.33

a. The use of time values is only allowed in firmware versions of A.02.00 and later.

Programming in C Using the VTL

The programming examples that are provided are written using the C programming language and the Agilent VTL (VISA transition library). This section includes some basic information about programming in the C language. Note that some of this information may not be relevant to your particular application. (For example, if you are not using VXI instruments, the VXI references will not be relevant).

Refer to your C programming language documentation for more details. (This information is taken from the manual "VISA Transition Library", part number E2090-90026.) The following topics are included:

Typical Example Program Contents

The following is a summary of the VTL function calls used in the example programs.

`visa.h` This file is included at the beginning of the file to provide the function prototypes and constants defined by VTL.

`ViSession` The `ViSession` is a VTL data type. Each object that will establish a communication channel must be defined as `ViSession`.

`viOpenDefaultRM` You must first open a session with the default resource manager with the `viOpenDefaultRM` function. This function will initialize the default resource manager and return a pointer to that resource manager session.

`viOpen` This function establishes a communication channel with the device specified. A session identifier that can be used with other VTL functions is returned. This call must be made for each device you will be using.

`viPrintf`
`viScanf` These are the VTL formatted I/O functions that are patterned after those used in the C programming language. The `viPrintf` call sends the IEEE 488.2 `*RST` command to the instrument and puts it in a known state. The `viPrintf` call is used again to query

for the device identification (`*IDN?`). The `viScanf` call is then used to read the results.

`viClose` This function must be used to close each session. When you close a device session, all data structures that had been allocated for the session will be de-allocated. When you close the default manager session, all sessions opened using the default manager session will be closed.

Linking to VTL Libraries

Your application must link to one of the VTL import libraries:

32-bit Version:

`C:\VXIPNP\WIN95\LIB\MSC\VISA32.LIB` for Microsoft compilers

`C:\VXIPNP\WIN95\LIB\BC\VISA32.LIB` for Borland compilers

16-bit Version:

`C:\VXIPNP\WIN\LIB\MSC\VISA.LIB` for Microsoft compilers

`C:\VXIPNP\WIN\LIB\BC\VISA.LIB` for Borland compilers

See the following section, "Compiling and Linking a VTL Program" for information on how to use the VTL run-time libraries.

Compiling and Linking a VTL Program

32-bit Applications

The following is a summary of important compiler-specific considerations for several C/C++ compiler products when developing WIN32 applications.

For Microsoft Visual C++ version 2.0 compilers:

- Select `Project | Update All Dependencies` from the menu.
- Select `Project | Settings` from the menu. Click on the `C/C++` button. Select `Code Generation` from the `Use Run-Time Libraries` list box. VTL requires these definitions for WIN32. Click `OK` to close the dialog boxes.
- Select `Project | Settings` from the menu. Click on the `Link` button and add `visa32.lib` to the `Object / Library Modules` list box. Optionally, you may add the library directly to your project file. Click `OK` to close the dialog boxes.
- You may wish to add the include file and library file search paths. They are set by doing the following:

  1. Select `Tools | Options` from the menu.
  2. Click `Directories` to set the include file path.

3. Select `Include Files` from the `Show Directories For` list box.

4. Click `Add` and type in the following:
   `C:\VXIPNP\WIN95\INCLUDE`

5. Select `Library Files` from the `Show Directories For` list box.

6. Click `Add` and type in the following:
   `C:\VXIPNP\WIN95\LIB\MSC`

For Borland C++ version 4.0 compilers:

- You may wish to add the include file and library file search paths. They are set under the `Options | Project` menu selection. Double-click on `Directories` from the `Topics` list box and add the following:

```
C:\VXIPNP\WIN95\INCLUDE
C:\VXIPNP\WIN95\LIB\BC
```

16-bit Applications

The following is a summary of important compiler-specific considerations for the Windows compiler.

For Microsoft Visual C++ version 1.5:

- To set the memory model, do the following:

  1. Select `Options | Project`.
  2. Click `Compiler`, then select `Memory Model` from the `Category` list.
  3. Click the `Model` list arrow to display the model options, and select `Large`.
  4. Click `OK` to close the `Compiler` dialog box.

- You may wish to add the include file and library file search paths. They are set under the `Options | Directories` menu selection:

```
C:\VXIPNP\WIN\INCLUDE
C:\VXIPNP\WIN\LIB\MSC
```

  Otherwise, the library and include files should be explicitly specified in the project file.

Example Program

This example program queries a GPIB device for an identification string and prints the results. Note that you must change the address.

```
/*idn.c - program filename */

#include "visa.h"
```

Programming Examples

```
#include <stdio.h>

void main ()
{
     /*Open session to GPIB device at address 18 */
     ViOpenDefaultRM (&defaultRM);
     ViOpen (defaultRM, GPIB0::18::INSTR", VI_NULL,
       VI_NULL, &vi);

     /*Initialize device */
     viPrintf (vi, "*RST\n");

     /*Send an *IDN? string to the device */
     printf (vi, "*IDN?\n");

     /*Read results */
     viScanf (vi, "%t", &buf);

     /*Print results */
     printf ("Instrument identification string: %s\n", buf);

     /* Close sessions */
     viClose (vi);
     viClose (defaultRM);
}
```


Including the VISA Declarations File

For C and C++ programs, you must include the visa.h header file at the beginning of every file that contains VTL function calls:

```
#include “visa.h”
```

This header file contains the VISA function prototypes and the definitions for all VISA constants and error codes. The visa.h header file includes the visatype.h header file.

The visatype.h header file defines most of the VISA types. The VISA types are used throughout VTL to specify data types used in the functions. For example, the viOpenDefaultRM function requires a pointer to a parameter of type ViSession. If you find ViSession in the visatype.h header file, you will find that ViSession is eventually typed as an unsigned long.

Opening a Session

A session is a channel of communication. Sessions must first be opened on the default resource manager, and then for each device you will be using. The following is a summary of sessions that can be opened:

- A **resource manager session** is used to initialize the VISA system. It is a parent session that knows about all the opened sessions. A resource manager session must be opened before any other session can be opened.

- A **device session** is used to communicate with a device on an interface. A device session must be opened for each device you will be using. When you use a device session you can communicate without worrying about the type of interface to which it is connected. This insulation makes applications more robust and portable across interfaces. Typically a device is an instrument, but could be a computer, a plotter, or a printer.

NOTE All devices that you will be using need to be connected and in working condition prior to the first VTL function call (`viOpenDefaultRM`). The system is configured only on the *first* `viOpenDefaultRM` per process. Therefore, if `viOpenDefaultRM` is called without devices connected and then called again when devices are connected, the devices will not be recognized. You must close **ALL** resource manager sessions and re-open with all devices connected and in working condition.

Device Sessions

There are two parts to opening a communications session with a specific device. First you must open a session to the default resource manager with the `viOpenDefaultRM` function. The first call to this function initializes the default resource manager and returns a session to that resource manager session. You only need to open the default manager session once. However, subsequent calls to `viOpenDefaultRM` returns a session to a unique session to the same default resource manager resource.

Next, you open a session with a specific device with the `viOpen` function. This function uses the session returned from `viOpenDefaultRM` and returns its own session to identify the device session. The following shows the function syntax:

viOpenDefaultRM (*sesn*);

viOpen (*sesn*, *rsrcName*, *accessMode*, *timeout*, *vi*);

The session returned from `viOpenDefaultRM` must be used in the *sesn* parameter of the `viOpen` function. The `viOpen` function then uses that session and the device address specified in the *rsrcName* parameter to open a device session. The *vi* parameter in `viOpen` returns a session identifier that can be used with other VTL functions.

Your program may have several sessions open at the same time by creating multiple session identifiers by calling the `viOpen` function multiple times.

The following summarizes the parameters in the previous function calls:

sesn This is a session returned from the `viOpenDefaultRM` function that identifies the resource manager session.

Programming Examples

rsrcName	This is a unique symbolic name of the device (device address).
accessMode	This parameter is not used for VTL. Use VI_NULL.
timeout	This parameter is not used for VTL. Use VI_NULL.
vi	This is a pointer to the session identifier for this particular device session. This pointer will be used to identify this device session when using other VTL functions.

The following is an example of opening sessions with a GPIB multimeter and a GPIB-VXI scanner:

```
ViSession defaultRM, dmm, scanner;
.
.
viOpenDefaultRM(&defaultRM);
viOpen (defaultRM, "GPIB0::22::INSTR", VI_NULL,
    VI_NULL, &dmm);
viOpen (defaultRM, "GPIB-VXI0::24::INSTR", VI_NULL,
    VI_NULL, &scanner);
.
.
viClose (scanner);
viClose (dmm);
viClose(defaultRM);
```

The above function first opens a session with the default resource manager. The session returned from the resource manager and a device address is then used to open a session with the GPIB device at address 22. That session will now be identified as **dmm** when using other VTL functions. The session returned from the resource manager is then used again with another device address to open a session with the GPIB-VXI device at primary address 9 and VXI logical address 24. That session will now be identified as **scanner** when using other VTL functions. See the following section for information on addressing particular devices.

Addressing a Session

As seen in the previous section, the *rsrcName* parameter in the `viOpen` function is used to identify a specific device. This parameter is made up of the VTL interface name and the device address. The interface name is determined when you run the VTL Configuration Utility. This name is usually the interface type followed by a number. The following table illustrates the format of the *rsrcName* for the different interface types:

Interface	**Syntax**
VXI	VXI [*board*]::*VXI logical address*[::INSTR]
GPIB-VXI	GPIB-VXI [*board*]::*VXI logical address*[::INSTR]

Interface	Syntax
GPIB	GPIB [*board*]::*primary address*[::*secondary address*][::INSTR]

The following describes the parameters used above:

board — This optional parameter is used if you have more than one interface of the same type. The default value for *board* is 0.

VSI logical address — This is the logical address of the VXI instrument.

primary address — This is the primary address of the GPIB device.

secondary address — This optional parameter is the secondary address of the GPIB device. If no secondary address is specified, none is assumed.

INSTR — This is an optional parameter that indicates that you are communicating with a resource that is of type **INSTR**, meaning instrument.

NOTE If you want to be compatible with future releases of VTL and VISA, you must include the INSTR parameter in the syntax.

The following are examples of valid symbolic names:

XI0::24::INSTR — Device at VXI logical address 24 that is of VISA type INSTR.

VXI2::128 — Device at VXI logical address 128, in the third VXI system (VXI2).

GPIB-VXI0::24 — A VXI device at logical address 24. This VXI device is connected via a GPIB-VXI command module.

GPIB0::7::0 — A GPIB device at primary address 7 and secondary address 0 on the GPIB interface.

The following is an example of opening a device session with the GPIB device at primary address23.

```
ViSession defaultRM, vi;

.
.

viOpenDefaultRM (&defaultRM);

viOpen (defaultRM, "GPIB0::23::INSTR", VI_NULL,VI_NULL,&vi);

.
.
```

```
viClose(vi);

viClose (defaultRM);
```

Closing a Session

The `viClose` function must be used to close each session. You can close the specific device session, which will free all data structures that had been allocated for the session. If you close the default resource manager session, all sessions opened using that resource manager will be closed.

Since system resources are also used when searching for resources (`viFindRsrc`) or waiting for events (`viWaitOnEvent`), the `viClose` function needs to be called to free up find lists and event contexts.

Index

Index

Index

Index

Index