User's Guide

Agilent Technologies
PSG Signal Generators

This guide applies to the following signal generator models:

E8267C PSG Vector Signal Generator
E8257C PSG Analog Signal Generator
E8247C PSG CW Signal Generator

Due to our continuing efforts to improve our products through firmware and hardware revisions, signal generator design and operation may vary from descriptions in this guide. We recommend that you use the latest revision of this guide to ensure you have up-to-date product information. Compare the print date of this guide (see bottom of page) with the latest revision, which can be downloaded from the following website:

www.agilent.com/find/psg

Agilent Technologies

Notice

The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions.

Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied with regard to this manual and to any of the Agilent products to which it pertains, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or any of the Agilent products to which it pertains. Should Agilent have a written contract with the User and should any of the contract terms conflict with these terms, the contract terms shall control.

Questions or Comments about our Documentation?

We welcome any questions or comments you may have about our documentation. Please send us an E-mail at **sources_manuals@am.exch.agilent.com**.

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

1   Signal Generator Overview

This chapter describes the models, options, and features available for Agilent PSG signal generators. The modes of operation, front panel user interface, as well as front and rear panel connectors are also described.

This chapter includes the following major sections:

- "Signal Generator Models and Features" on page 2
- "Options" on page 5
- "Modes of Operation" on page 10
- "Front Panel" on page 11
- "Front Panel Display" on page 19
- "Rear Panel" on page 24

Signal Generator Models and Features

Table 1-1 lists the available PSG signal generator models along with their output signal types and frequency range.

Table 1-1   **PSG Signal Generator Models**

Model	Type	Frequency Range
E8247C PSG CW signal generator	CW	250 kHz to 20 GHz, or 250 kHz to 40 GHz
E8257C PSG analog signal generator	Analog	250 kHz to 20 GHz, or 250 kHz to 40 GHz
E8267C PSG vector signal generator	Vector	250 kHz to 20 GHz

E8247C PSG CW Signal Generator Features

The E8247C PSG CW signal generator includes the following features:

- CW output from 250 kHz to 20 GHz or 40 GHz

- frequency resolution to 0.001 Hz

- list and step sweep of frequency and amplitude, with multiple trigger sources

- user flatness correction

- external diode detector leveling

- automatic leveling control (ALC) on and off modes; power calibration in ALC-off mode is available, even without power search

- 10 MHz reference oscillator with external output

- RS-232, GPIB, and 10Base-T LAN I/O interfaces

- a millimeter head interface that is compatible with Agilent 83550 Series millimeter heads (for frequency extension up to 110 GHz)

E8257C PSG Analog Signal Generator Features

The E8257C PSG analog signal generator provides all the functionality
of the E8247C PSG CW signal generator and adds the following features:

- open-loop or closed-loop AM

- dc-synthesized FM to 10 MHz rates; maximum deviation depends on the carrier frequency

- phase modulation (ΦM)

- pulse modulation

- external modulation inputs for AM, FM, ΦM, and pulse

- simultaneous modulation configurations (except: FM with ΦM or Linear AM with Exponential AM)

- an internal pulse generator that includes the following:

  — selectable pulse modes: internal square, internal free-run, internal triggered, internal doublet, internal
    gated, and external pulse; internal triggered, internal doublet, and internal gated require an
    external trigger source

  — adjustable pulse rate

  — adjustable pulse period

  — adjustable pulse width

  — adjustable pulse delay

  — selectable external pulse triggering: positive or negative

- dual function generators that includes the following:

  — 50Ω low frequency output, 0 to 3 V_p, available through LF OUTPUT

  — selectable waveforms: sine, dual-sine, swept-sine, triangle, positive ramp, negative ramp, square,
    uniform noise, Gaussian noise, and dc

  — adjustable frequency modulation rates

  — selectable triggering in list and step sweep modes: free run (auto), trigger key (single), bus (remote),
    and external

E8267C PSG Vector Signal Generator Features

The E8267C PSG vector signal generator provides all the functionality
of the E8257C PSG analog signal generator and adds the following features:

- internal I/Q modulator

- external analog I/Q inputs

- single-ended and differential analog I/Q outputs

Options

Table 1-2, Table 1-3, and Table 1-4 show the available hardware and accessory options for the PSG signal generators.

Table 1-2   **E8247C PSG CW Signal Generator Hardware Options**

Options	Description
520	250 kHz to 20 GHz frequency coverage
540	250 kHz to 40 GHz frequency coverage
007	add ramp sweep and the following measurements can be made: • Perform frequency sweeps to determine the frequency response, power level accuracy, and flatness of a device under test. • Perform power sweeps to measure an amplifier's saturation level and determine its 1 dB compression point.
ABA	add user documentation set, English language (paper manuals)
CD1	add user documentation CD-ROM, English language (CD-ROM containing .pdf files) One copy of the documentation CD-ROM is always included with each PSG order.
0BW	add service guide, assembly level (paper manual)
1CM	add rack-mount kit
1CN	add handle kit
1CP	add rack-mount kit and handle kit
1E1	add 115 dB mechanical step attenuator
1EA	add high RF output power
1ED	add Type-N RF output connector (in place of APC 3.5 mm connector); for use on Option 520 models only
1EM	move all front panel connectors to the rear panel
UK6	add commercial calibration certificate with test data
UNR	add enhanced close-in phase noise

Table 1-3   **E8257C PSG Analog Signal Generator Hardware Options**

Options	Description
520	250 kHz to 20 GHz frequency coverage
540	250 kHz to 40 GHz frequency coverage
007	add ramp sweep and the following measurements can be made: • Perform frequency sweeps to determine the frequency response, power level accuracy, and flatness of a device under test. • Perform power sweeps to measure an amplifier's saturation level and determine its 1 dB compression point.
ABA	add user documentation set, English language (paper manuals)
CD1	add user documentation set, English language (CD containing .pdf files) One copy of CD1 is included with each order for free.
0BW	add service guide, assembly level (paper manual)
1CM	add rack-mount kit
1CN	add handle kit
1CP	add rack-mount kit and handle kit
1E1	add 115 dB mechanical step attenuator
1E6	add narrow pulse modulation (500 MHz – 3.2 GHz)
1EA	add high RF output power
1ED	add Type-N RF output connector (in place of APC 3.5 mm connector); for use on Option 520 models only
1EM	move all front panel connectors to the rear panel
UK6	add commercial calibration certificate with test data
UNR	add enhanced close-in phase noise

Table 1-4   **E8267C PSG Vector Signal Generator Hardware Options**

Options[a]	Description
520	250 kHz to 20 GHz frequency coverage
002/602	add internal baseband generator
	Option 002: 32 megasample memory (no longer available) Option 602: 64 megasample memory with digital bus capability
	The following modes of operation are available:
	• Dual-Arb mode allows you to download waveform files through the RS-232, GPIB, or LAN into the internal baseband generator and play them
	• Two Tone mode is a personality that allows you to build a waveform that has two equal-powered channels, or tones. The default waveform has two tones that are symmetrically spaced from the center carrier frequency and have user-defined amplitude, carrier frequency, and frequency separation settings. You can also align the tones left or right, relative to the carrier frequency. Two-tone waveforms are created using the internal baseband generator and stored in memory for playback.
	• Multitone mode is a personality that allows you to build a waveform that has up to 64 channels, or tones. Using the `Multitone Setup` table editor you can define, modify, and store waveforms for playback. Multitone waveforms are created using the internal baseband generator and stored in memory for playback.
	• Custom mode allows both real time and arbitrary waveforms to be created
	— real time waveforms can be created by supplying random data
	— arbitrary waveforms can be created and repeated using the internal baseband generator (these waveforms could also be generated externally and downloaded to the internal baseband generator)
003	Enables ESG digital output connectivity with N5102A (Option 602 required)
005	add internal hard drive (6 GB non-volatile waveform storage)
007	add ramp sweep and the following measurements can be made:
	• Perform frequency sweeps to determine the frequency response, power level accuracy, and flatness of a device under test.
	• Perform power sweeps to measure an amplifier's saturation level and determine its 1 dB compression point.

Table 1-4   **E8267C PSG Vector Signal Generator Hardware Options**

Options[a]	Description
015	add wideband external I/Q inputs Provides up to 1 GHz uncalibrated RF modulation bandwidth. Standard external I/Q inputs provide 160 MHz RF modulation bandwidth.
ABA	add user documentation set, English language (paper manuals)
CD1	add user documentation set, English language (CD containing .pdf files) One copy of CD1 is included with each order for free.
0BW	add service guide, assembly level (paper manual)
1CM	add rack-mount kit
1CN	add handle kit
1CP	add rack-mount kit and handle kit
1E6	add narrow pulse modulation (500 MHz – 3.2 GHz)
1ED	add Type-N RF output connector (in place of APC 3.5 mm connector)
1EM	move all front panel connectors to the rear panel
UK6	add commercial calibration certificate with test data
UNR	add enhanced close-in phase noise

a. The functionality of both Option 1E1 (add 115 dB mechanical step attenuator) and Option 1EA (add high RF output power) are included as standard features in the E8267C PSG vector signal generator.

Table 1-5   **Signal Studio Software Personalities**

Option	Description
408	Enhanced Multitone
420	Pulse Building
421	Noise Power Ratio
This listing of Signal Studio software personalities may not be complete. For a current list of personalities, refer to **www.agilent.com/find/signalstudio**.	

Understanding E8267C Baseband Generator Options

Your E8267C PSG can have one of two baseband generator options, depending upon when you first purchased the instrument and whether or not the baseband generator has been upgraded to a newer version. The options consist of the following:

Option 002   internal baseband generator with 32 megasample memory (no longer available)

Option 602   internal baseband generator with 64 megasample memory and digital bus capability

These two baseband generators are similar in some respects, and very different in others. They are similar in that both versions support the same signal generation formats, such as Custom Real-Time Baseband I/Q, Multitone, and others. Both versions offer arbitrary waveform capability and real-time baseband generation capability.

The first obvious difference between the baseband generators is that they offer different playback memory sizes. This is primarily important to users providing their own arbitrary waveform files, since these may be very large in size.

The second important difference between these baseband generators is that only Option 602 offers digital bus capability. This proprietary digital bus is used for communication with the Agilent Baseband Studio suite of products. Baseband Studio products provide a range of baseband signal processing functions, including baseband digital outputs and hard drive waveform streaming. These functions are not compatible with Option 002. An upgrade kit (E8267K-602) is available, however, if you want to add Option 602 to your existing E8267C PSG.

Modes of Operation

PSG signal generator models can be used in CW mode:

- CW mode produces a single carrier signal.

    — If you have an E8247C PSG CW signal generator, you can produce a CW single carrier signal
    without modulation.

    — If you have an E8257C PSG analog signal generator, you can produce a CW single carrier signal
    without modulation or you can add AM, FM, ΦM, or Pulse modulation to produce a single carrier
    modulated signal; some of these modulations can be used together.

    — If you have an E8267C PSG vector signal generator, you can produce a CW single carrier signal
    without modulation or you can add AM, FM, ΦM, Pulse, or I/Q modulation to produce a single
    carrier modulated signal; some of these modulations can be used together.

In addition to CW and analog mode, all of the following modes are also available to the E8267C PSG vector
signal generator:

- Custom Arb Waveform Generator mode can produce a single modulated carrier or multiple modulated
  carriers. Each modulated carrier waveform must be calculated and generated before it can be output; this
  signal generation occurs on the internal baseband generator (Option 002/602). Once a waveform has
  been created, it can be stored and recalled which enables repeatable playback of test signals. To learn
  more, refer to "Custom Arb Waveform Generator" on page 103.

- Custom Real Time I/Q Baseband mode produces a single carrier, but it can be modulated with real time
  data that allows real time control over all of the parameters that affect the signal. The single carrier signal
  that is produced can be modified by applying various data patterns, filters, symbol rates, modulation
  types, and burst shapes. To learn more, refer to "Custom Real Time I/Q Baseband" on page 129.

- Two Tone mode produces two separate carrier signals without any kind of modulation; the frequency
  spacing between the two carrier signals is adjustable as well as the amplitude of both carriers. To learn
  more, refer to "Two-Tone Waveform Generator" on page 223.

- Multitone mode produces any number of carrier signals without any kind of modulation; like Two Tone
  mode, the frequency spacing between all carrier signals is adjustable as well as the amplitude of all
  carriers. To learn more, refer to "Multitone Waveform Generator" on page 213.

- Dual ARB mode is used to control the playback sequence of waveform segments that have been written
  into the ARB memory located on the internal baseband generator (Option 002/602). These waveforms
  can be generated using the internal baseband generator, in Custom Arb Waveform Generator mode, or
  downloaded through a remote interface into the ARB memory. To learn more, refer to "Dual Arbitrary
  Waveform Generator" on page 181.

Front Panel

Figure 1-1 shows the E8267C PSG vector signal generator front panel with a list of items called out that enable you to define, monitor, and manage input and output characteristics.

The description of each item also applies to both the E8257C PSG analog signal generator and the E8247C PSG CW signal generator front panels. Not all items being described are available on every signal generator; the list of items that your particular signal generator has is dependent on its model and options.

Figure 1-1   **Front Panel Diagram (E8267C PSG Vector Signal Generator)**
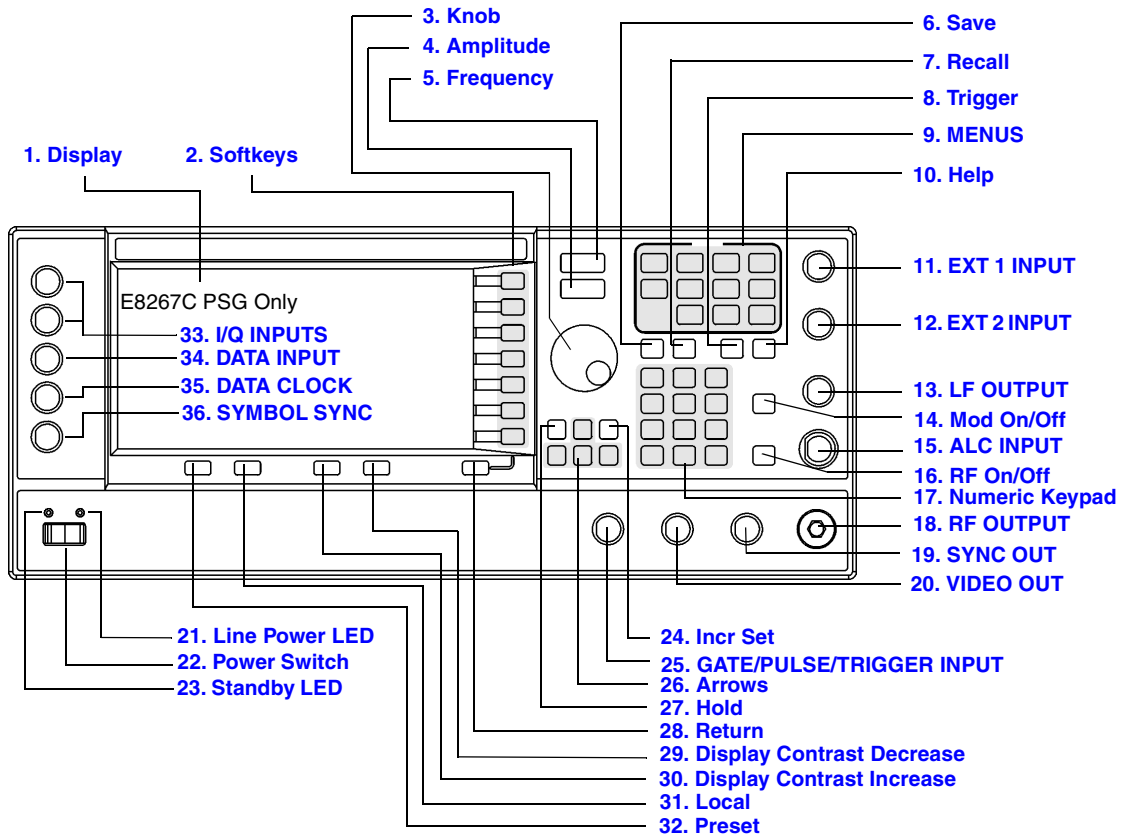
1. Display

The LCD screen provides information on the current function. Information can include status indicators, frequency and amplitude settings, and error messages. Softkeys labels are located on the right-hand side of the display. For further descriptions of the front panel display, refer to "Front Panel Display" on page 19.

2. Softkeys

Softkeys activate the displayed function to the left of each key.

3. Knob

Use the knob to increase or decrease a numeric value, changes a highlighted digit or character, or step through lists or select items in a row.

4. Amplitude

Pressing this hardkey makes amplitude the active function. You can change the output amplitude or use the menus to configure amplitude attributes such as power search, user flatness, and leveling mode.

5. Frequency

Pressing this hardkey makes frequency the active function. You can change the output frequency or use the menus to configure frequency attributes such as frequency multiplier, offset, and reference.

6. Save

Pressing this hardkey accesses a menu of choices enabling you to save data in the instrument state register. The instrument state register is a section of memory divided into 10 sequences (numbered 0 through 9) each containing 100 registers (numbered 00 through 99).

It is used to store and recall:

*   frequency and amplitude settings on an E8247C PSG CW signal generator
*   frequency, amplitude, and modulation settings on an E8257C PSG analog signal generator or E8267C PSG vector signal generator

The Save hardkey provides a quick alternative to reconfiguring the signal generator through the front panel or SCPI commands when switching between different signal configurations. Once an instrument state has been saved, all of the frequency, amplitude, and modulation settings can be recalled with the **Recall** hardkey.

7. Recall

Restores an instrument state saved in a memory register. Refer to the **Save** hardkey for further information.

8. Trigger

Initiates an immediate trigger event for a function such as a list, step, or ramp sweep (Option 007 only).

Before this hardkey can be used to initiate a trigger event, the trigger mode must be set to **Trigger Key**. For example: press the Sweep/List hardkey, then one of the following sequences of softkeys:

- **More (1 of 2) > Sweep Trigger > Trigger Key**
- **More (1 of 2) > Point Trigger > Trigger Key**

9. MENUS

These keys open softkey menus for configuring various functions. For descriptions, see the *Key Reference*.

Table 1-6   **Hardkeys in MENUS Group of Front Panel**

E8247C PSG CW	E8257C PSG Analog	E8267C PSG Vector	
Sweep/List Utility	AM Sweep/List FM/ΦM Utility Pulse LF Out	Mode Mux AM Sweep/List Mode Setup Aux Fctn	FM/Φ M Utility I/Q Pulse LF Out

10. Help

Pressing this hardkey accesses a short description of any hardkey or softkey. There are two help modes available on the signal generator: single and continuous. The single mode is the factory preset condition. Toggle between single and continuous mode by pressing **Utility > Instrument Info/Help Mode > Help Mode Single Cont**.

- In single mode, help text is provided for the next key you press without activating the key's function. Any key pressed afterward exits the help mode and its function is activated.

- In continuous mode, help text is provided for each subsequent key press until you press the **Help** hardkey again or change to single mode. In addition, each key is active, meaning that the key function is executed (except for the Preset key).

11. EXT 1 INPUT

This female BNC input connector (E8257C PSG and E8267C PSG only)
accepts a ± 1 V_p signal for AM, FM, and ΦM. For all these modulations, ± 1 V_p produces the indicated
deviation or depth.

When ac-coupled inputs are selected for AM, FM, or ΦM and the peak input voltage differs from 1 V_p by
more than 3%, the HI/LO annunciators light on the display. The input impedance is selectable as either 50Ω
or 600Ω and the damage levels are 5 V_{rms} and 10 V_p.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

12. EXT 2 INPUT

This female BNC input connector (E8257C PSG and E8267C PSG only)
accepts a ± 1 V_p signal for AM, FM, and ΦM. With AM, FM, or ΦM, ± 1 V_p produces the indicated deviation
or depth.

When ac-coupled inputs are selected for AM, FM, or ΦM and the peak input voltage differs from 1 V_p by
more than 3%, the HI/LO annunciators light on the display. The input impedance is selectable as either 50Ω
or 600Ω and damage levels are 5 V_{rms} and 10 V_p.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

13. LF OUTPUT

This female BNC output connector (E8257C PSG and E8267C PSG only) is the output for modulation
signals generated by the low frequency (LF) source function generator. This output is capable of driving
3 V_p (nominal) into a 50Ω load.

On signal generators with Option 1EM, this output is relocated to a rear panel female BNC connector.

14. Mod On/Off

Pressing this hardkey (E8257C PSG and E8267C PSG only)
enables or disables all active modulation formats (AM, FM, ΦM, Pulse, or I/Q) that are applied to the output
carrier signal available through the RF Output connector.

This hardkey does not set up or activate an AM, FM, ΦM, Pulse, or I/Q format; each individual modulation
format must still be set up and activated (for example, **AM > AM On**) or nothing will be applied to the output
carrier signal when the **Mod On/Off** hardkey is enabled.

The MOD ON/OFF annunciator, which is always present on the display, indicates whether active modulation
formats have been enabled or disabled with the **Mod On/Off** hardkey.

15. ALC INPUT

This female BNC input connector is used for negative external detector leveling. This connector accepts an input of −0.2 mV to −0.5 V. The nominal input impedance is 120 kΩ and the damage level is ±15 V.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

16. RF On/Off

Pressing this hardkey toggles the operating state of the RF signal present at the RF OUTPUT connector. Although you can set up and enable various frequency, power, and modulation states, the RF and microwave output signal is not present at the RF OUTPUT until **RF On/Off** is set to On. An annunciator is always visible in the display to indicate whether the RF is turned on or off.

17. Numeric Keypad

The numeric keypad consists of the 0 through 9 hardkeys, a decimal point hardkey, and a backspace hardkey ($\boxed{+/\!-\!\leftarrow}$). The backspace hardkey enables you to backspace or alternate between a positive and a negative value. When specifying a negative numeric value, the negative sign must be entered prior to entering the numeric value.

18. RF OUTPUT

This connector is the output for RF and microwave signals. The nominal output impedance is 50Ω. The reverse-power damage levels are 0 Vdc, 0.5 watts nominal.

On signal generators with Option 1EM, this output is relocated to a rear panel female BNC connector.

19. SYNC OUT

This female BNC output connector (E8257C PSG and E8267C PSG only) outputs a synchronizing TTL-compatible pulse signal that is nominally 50 ns wide during internal and triggered pulse modulation. The nominal source impedance is 50Ω.

On signal generators with Option 1EM, this output is relocated to a rear panel female BNC connector.

20. VIDEO OUT

This female BNC output connector (E8257C PSG and E8267C PSG only) outputs a TTL-level compatible pulse signal that follows the output envelope in all pulse modes. The nominal source impedance is 50Ω.

On signal generators with Option 1EM, this output is relocated to a rear panel female BNC connector.

21. Line Power LED

This green LED indicates when the signal generator power switch is set to the on position.

22. Power Switch

This switch activates full power to the signal generator when set to the on position and deactivates all signal generator functions when in standby mode. In standby mode, the signal generator remains connected to the line power and power is supplied to some internal circuits.

23. Standby LED

This yellow LED indicates when the signal generator power switch is set to the standby condition.

24. Incr Set

Pressing this hardkey enables you to set the increment value of the current active function. When this hardkey is pressed, the increment value of the current active function will appear in the active entry area of the display. Use the numeric keypad, arrow hardkeys, or the knob to adjust the increment value.

25. GATE/PULSE/TRIGGER INPUT

This female BNC input connector (E8257C PSG and E8267C PSG only)
accepts an externally supplied pulse signal for use as a pulse or trigger input. With pulse modulation, +1 V is on and 0 V is off (trigger threshold of 0.5 V with a hysteresis of 10%; so 0.6 V would be on and 0.4 V would be off). The damage levels are ± 5 V_{rms} and 10 V_p. The nominal input impedance is 50Ω.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

26. Arrows

These up and down arrow hardkeys are used to increase or decrease a numeric value, step through displayed lists, or to select items in a row of a displayed list. Individual digits or characters may be highlighted using the left and right arrow hardkeys. Once an individual digit or character is highlighted, its value can be changed using the up and down arrow hardkeys.

27. Hold

Pressing this hardkey blanks the softkey label area and text areas on the display. Softkeys, arrow hardkeys, the knob, the numeric keypad, and the **Incr Set** hardkey have no effect once this hardkey is pressed.

28. Return

Pressing this hardkey will return the signal generator one level back from its current softkey menu level to the previous softkey menu level. It enables you to step back through the menus until you reach the first menu you selected.

29. Display Contrast Decrease

Pressing this hardkey causes the display background to darken.

30. Display Contrast Increase

Pressing this hardkey causes the display background to lighten.

31. Local

Pressing this hardkey deactivates remote operation and returns the signal generator to front panel control.

32. Preset

Pressing this hardkey sets the signal generator to a known state (factory or user-defined).

33. I/Q INPUTS

These female BNC input connectors (E8267C PSG only) accept an externally supplied, analog, I/Q modulation; the in-phase component is supplied through the I INPUT and the quadrature-phase component is supplied through the Q INPUT. The signal level is

$\sqrt{I^2+Q^2}$ = 0.5 V_{rms} for a calibrated output level. The nominal input impedance is 50Ω or 600Ω. The damage level is 1 V_{rms} and 10 V_{peak}.

To activate signals applied to these connectors, press **Mux > I/Q Source 1** or **I/Q Source 2** and then select either **Ext 50 Ohm** or **Ext 600 Ohm**.On signal generators with Option 1EM, these inputs are relocated to rear panel female BNC connectors.

34. DATA INPUT

This female BNC input connector (E8267C PSG with Option 002/602 only) is CMOS compatible and accepts an externally supplied serial data input for digital modulation applications. The expected input is a 3.3 V CMOS signal (which is also TTL compatible) where a CMOS high is equivalent to a data 1 and a CMOS low is equivalent to a data 0. The maximum input data rate is 50 Mb/s. The data must be valid on the falling edges of the data clock (normal mode) or the on the falling edges of the symbol sync (symbol mode). The damage levels are > +5.5 and < −0.5 V.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

35. DATA CLOCK INPUT

This female BNC input connector (E8267C PSG only) is CMOS compatible and accepts an externally supplied data-clock input signal to synchronize serial data for use with the internal baseband generator (Option 002/602). The expected input is a 3.3 V CMOS bit clock signal (which is also TTL compatible) where the rising edge is aligned with the beginning data bit. The falling edge is used to clock the DATA and SYMBOL SYNC signals. The maximum clock rate is 50 MHz. The damage levels are > +5.5 and < −0.5 V.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

36. SYMBOL SYNC INPUT

This female BNC input connector (E8267C PSG only) is CMOS compatible and accepts an externally supplied symbol sync signal for use with the internal baseband generator (Option 002/602). The expected input is a 3.3 V CMOS bit clock signal (which is also TTL compatible). SYMBOL SYNC might occur once per symbol or be a single one-bit-wide pulse that is used to synchronize the first bit of the first symbol. The maximum clock rate is 50 MHz. The damage levels are > +5.5 and < −0.5 V.

SYMBOL SYNC may be used in two modes:

- When used as a symbol sync in conjunction with a data clock, the signal must be high during the first data bit of the symbol. The signal must be valid during the falling edge of the data clock signal and may be a single pulse or continuous.

- When the SYMBOL SYNC itself is used as the (symbol) clock, the CMOS falling edge is used to clock the DATA signal.

On signal generators with Option 1EM, this input is relocated to a rear panel female BNC connector.

Front Panel Display

Figure 1-2 shows the front panel display. The LCD screen displays data fields, annotations, key press results, softkey labels, error messages, and annunciators that represent various active signal generator functions.

Figure 1-2   Front Panel Display Diagram
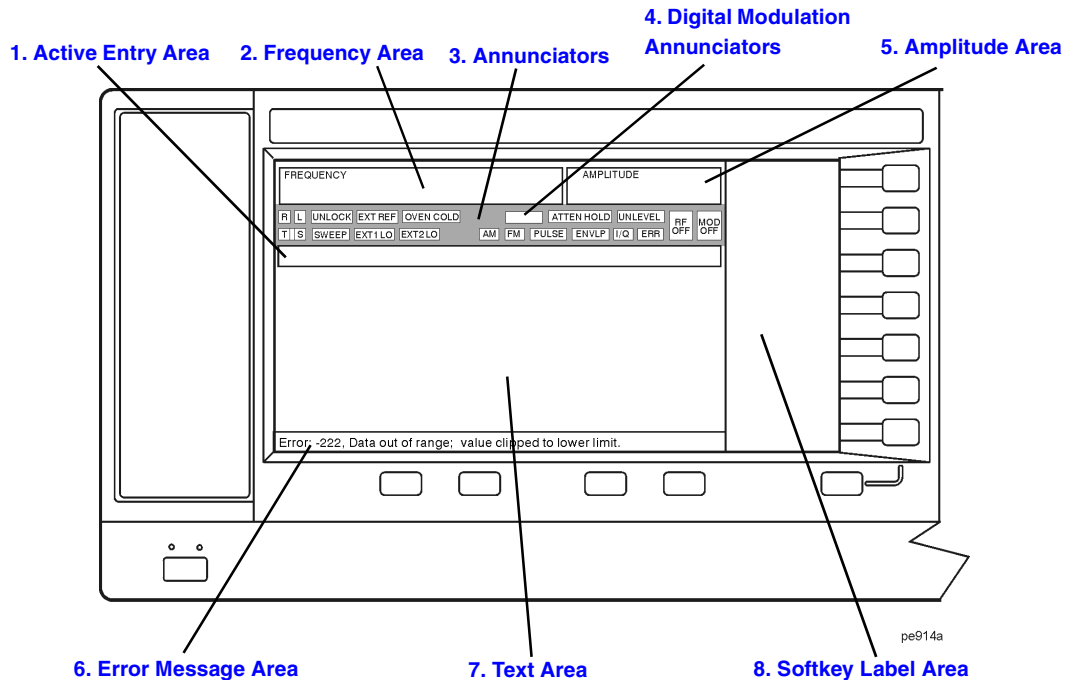
1. Active Entry Area

The current active function is shown in this area. For example, if frequency is the active function, the current frequency setting will be displayed here. If the current active function has an increment value associated with it, that value is also displayed.

2. Frequency Area

The current frequency setting is shown in this portion of the display. Indicators are also displayed in this area when the frequency offset or multiplier is used, the frequency reference mode is turned on, or a source module is enabled.

3. Annunciators

The display annunciators show the status of some of the signal generator functions and indicate any error conditions. An annunciator position may be used by more than one function. This does not create a problem, because only one function that shares an annunciator position can be active at a time.

ΦM	This annunciator (E8257C PSG and E8267C PSG only) appears when phase modulation is turned on. If frequency modulation is turned on, the FM annunciator will replace ΦM.
ALC OFF	This annunciator appears when the ALC circuit is disabled. A second annunciator, UNLEVEL, will appear in the same position if the ALC is enabled and is unable to maintain the output level.
AM	This annunciator (E8257C PSG and E8267C PSG only) appears when amplitude modulation is turned on.
ARMED	This annunciator appears when a sweep has been initiated and the signal generator is waiting for the sweep trigger event.
ATTEN HOLD	This annunciator (Option 1E1 or E8267C PSG only) appears when the attenuator hold function is turned on. When this function is on, the attenuator is held at its current setting.
ENVLP	This annunciator appears if a burst condition exists, such as when marker 2 is set to enable RF blanking in the Dual ARB personality.
ERR	This annunciator appears when an error message is placed in the error queue. This annunciator will not turn off until you have either viewed all of the error messages or cleared the error queue. You can access error messages by pressing **Utility > Error Info**.
EXT	This annunciator appears when external leveling is turned on.
EXT1 LO/HI	This annunciator (E8257C PSG and E8267C PSG only) is displayed as either EXT1 LO or EXT1 HI. This annunciator appears whenever the ac-coupled signal to the EXT 1 INPUT is less than 0.97 V_p or greater than 1.03 V_p.
EXT2 LO/HI	This annunciator (E8257C PSG and E8267C PSG only) is displayed as either EXT2 LO or EXT2 HI. This annunciator appears whenever the ac-coupled signal to the EXT 2 INPUT is less than 0.97 V_p or greater than 1.03 V_p.
EXT REF	This annunciator appears when an external frequency reference is applied.
FM	This annunciator (E8257C PSG and E8267C PSG only) appears when frequency modulation is turned on. If phase modulation is turned on, the ΦM annunciator will replace FM.
I/Q	This annunciator (E8267C PSG with Option 002/602 only) appears when I/Q modulation is turned on.

L	This annunciator appears when the signal generator is in listener mode and is receiving information or commands over the RS-232, GPIB, or VXI-11 LAN interface.
MOD ON/OFF	This annunciator (E8257C PSG and E8267C PSG only) which is always present on the display, indicates whether active modulation formats have been enabled or disabled with the **Mod On/Off** hardkey.
	Pressing the **Mod On/Off** hardkey enables or disables all active modulation formats (AM, FM, ΦM, Pulse, or I/Q) that are applied to the output carrier signal available through the RF Output connector.
	The **Mod On/Off** hardkey does not set up or activate an AM, FM, ΦM, Pulse, or I/Q format; each individual modulation format must still be set up and activated (for example, **AM > AM On**) or nothing will be applied to the output carrier signal when the **Mod On/Off** hardkey is enabled.
OVEN COLD	This annunciator (Option UNR only) appears when the temperature of the internal oven reference oscillator has dropped below an acceptable level. When this annunciator is on, frequency accuracy is degraded. This condition should occur only if the signal generator is disconnected from line power.
PULSE	This annunciator (E8257C PSG and E8267C PSG only) appears when pulse modulation is turned on.
R	This annunciator appears when the signal generator is remotely controlled over the GPIB, RS-232, or VXI-11/Sockets LAN interface (TELNET operation does not activate the R annunciator). When the R annunciator is on, the front panel keys are disabled, except for the Local key and the line power switch. For information on remote operation, refer to the *Programming Guide*.
RF ON/OFF	This annunciator indicates when the RF and microwave signal is present (RF ON) at the RF OUTPUT, or if the RF and microwave signal is not present (RF OFF) at the RF OUTPUT. Either condition of this annunciator is always visible in the display.
S	This annunciator appears when the signal generator has generated a service request (SRQ) over the RS-232, GPIB, or VXI-11 LAN interface.

SWEEP	This annunciator appears when the signal generator is in list, step, or ramp sweep mode; ramp sweep is available with Option 007 only.
	List mode is when the signal generator can jump from point to point in a list (hop list); the list is traversed in ascending or descending order. The list can be a frequency list, a power level list, or both.
	Step mode is when a start, stop, and step value (frequency or power level) are defined and the signal generator produces signals that start at the start value and increment by the step value until it reaches the stop value.
	Ramp sweep mode (Option 007 only) is when a start and stop value (frequency or power level) are defined and the signal generator produces signals that start at the start value and produce a continuous output until it reaches the stop value.
T	This annunciator appears when the signal generator is in talker mode and is transmitting information over the GPIB, RS-232, or VXI-11 LAN interface.
UNLEVEL	This annunciator appears when the signal generator is unable to maintain the correct output level. The UNLEVEL annunciator is not necessarily an indication of instrument failure. Unleveled conditions can occur during normal operation. A second annunciator, ALC OFF, will appear in the same position when the ALC circuit is disabled.
UNLOCK	This annunciator appears when any of the phase locked loops are unable to maintain phase lock. You can determine which loop is unlocked by examining the error messages.

4. Digital Modulation Annunciators

All digital modulation annunciators (E8267C PSG with Option 002/602 only) appear in this location. These annunciators appear only when the modulation is active, and only one digital modulation can be active at any given time.

ARB	Dual Arbitrary Waveform Generator
CUSTOM	Custom Real Time I/Q Baseband
DIGMOD	Custom Arb Waveform Generator
M-TONE	Multitone Waveform Generator
T-TONE	Two-Tone Waveform Generator

5. Amplitude Area

The current output power level setting is shown in this portion of the display. Indicators are also displayed in this area when amplitude offset is used, amplitude reference mode is turned on, external leveling mode is enabled, a source module is enabled, and when user flatness is enabled.

6. Error Message Area

Abbreviated error messages are reported in this space. When multiple error messages occur, only the most recent message remains displayed. Reported error messages with details can be viewed by pressing **Utility** > **Error Info**.

7. Text Area

This text area of the display is used for the following:

- show status information about the signal generator such as the modulation status, sweep lists, and file catalogs

- display the tables

- enables you to perform functions such as managing information, entering information, and displaying or deleting files

8. Softkey Label Area

The labels in this area define the function of the softkeys located immediately to the right of the label. The softkey label may change depending upon the function selected.

Rear Panel

The signal generator rear panel (Figure 1-3) provides input, output, and remote interface connections. Descriptions are provided for each rear panel connector. When Option 1EM is added, all front panel connectors are moved to the real panel; for a description of these connectors, see "Front Panel" on page 11.

Figure 1-3   **Rear Panel Diagram**

16. Digital Bus
15. AUXILIARY I/O
17. WIDEBAND I INPUT
18. WIDEBAND Q INPUT
19. COH
20. I OUT
21. I-bar OUT
22. Q OUT
23. Q-bar OUT
24. BASEBAND GEN
25. SMI
26. 10 MHz OUT
27. 10 MHz IN
28. 10 MHz EFC
(Option UNR)

e8267c_rear_panel

1. AC Power Receptacle
2. GPIB
3. AUXILIARY INTERFACE
4. LAN

14. BURST GATE IN
13. PATTERN TRIG IN
12. EVENT 2
11. EVENT 1

5. STOP SWEEP IN/OUT
6. Z-AXIS BLANK/MKRS
7. SWEEP OUT
8. TRIGGER OUT
9. TRIGGER IN
10. SOURCE SETTLED

1. AC Power Receptacle

The ac line voltage is connected here. The power cord receptacle accepts a three-pronged power cable that is shipped with the signal generator.

2. GPIB

This GPIB interface allows listen and talk capability with compatible IEEE 488.2 devices.

3. AUXILIARY INTERFACE

This 9-pin D-subminiature female connector is an RS-232 serial port that can be used for serial communication and Master/Slave source synchronization.

Table 1-7   **Auxiliary Interface Connector**

Pin Number	Signal Description	Signal Name
1	No Connection	
2	Receive Data	RECV
3	Transmit Data	XMIT
4	+5 V	
5	Ground, 0 V	
6	No Connection	
7	Request to Send	RTS
8	Clear to Send	CTS
9	No Connection	

Figure 1-4

5• 4• 3• 2• 1•
9• 8• 7• 6•

View looking into
rear panel connector
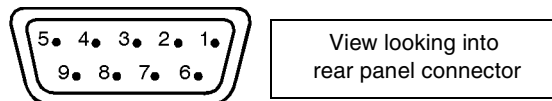
4. LAN

This LAN interface allows ethernet local area network communication through a 10Base-T LAN cable. The yellow LED on the interface illuminates when data transmission (transfer/receive) is present. The green LED illuminates when there is a delay in data transmission or no data transmission is present.

5. STOP SWEEP IN/OUT

This female BNC connector (Option 007 only) provides an open-collector, TTL-compatible, input/output signal that is used during ramp sweep operation. It provides low level (nominally 0 V) output during sweep retrace and band-cross intervals. It provides high level (nominally +5 V) output during the forward portion of sweep. Sweep will stop when this input/output connector is grounded externally.

6. Z-AXIS BLANK/MKRS

This female BNC connector (Option 007 only) supplies a +5 V (nominal) level during retrace and band-switch intervals of a step, list, or ramp sweep. During ramp sweep, this female BNC connector supplies a –5 V (nominal) level when the RF frequency is at a marker frequency and intensity marker mode is on.

This connection is most commonly used for interfacing with the Agilent 8757D scalar network analyzer.

7. SWEEP OUT

This female BNC connector outputs a voltage proportional to the RF power or frequency sweep ranging from 0 V at the start of sweep and goes to +10 V (nominal) at the end of sweep, regardless of sweep width.

The output impedance is less than 1Ω and can drive a 2 kΩ load.

When connected to an Agilent Technologies 8757D network analyzer, it generates a selectable number of equally spaced 1 ms 10 V pulses (nominal) across a ramp (analog) sweep. The number of pulses can be set from 101 to 1601 by remote control through the 8757D.

8. TRIGGER OUT

This female BNC connector, in step/list sweep mode, outputs a TTL signal that is high at the start of a dwell sequence or when waiting for a point trigger in manual sweep mode. The signal is low when the dwell is over or when a point trigger is received.

In ramp sweep mode, the output provides 1601 equally-spaced 1 µs pulses (nominal) across a ramp sweep. When using LF Out, the output provides a 2 µs pulse at the start of LF sweep.

9. TRIGGER IN

This female BNC connector accepts a TTL signal used for point-to-point triggering in manual sweep mode or a low-frequency (LF) sweep in external sweep mode. Triggering can occur on either the positive or negative edge of the TTL signal start.

The damage level is ≤ -4 V or $\geq +10$ V.

10. SOURCE SETTLED

This female BNC connector provides an indication when the signal generator has settled to a new frequency or power level. A low indicates that the source has settled.

11. EVENT 1

This female BNC connector (E8267C PSG only) is used with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this female BNC connector is non-functional.

In real-time mode, the EVENT 1 connector outputs a pattern or frame synchronization pulse for triggering or gating external equipment. It may be set to start at the beginning of a pattern, frame, or timeslot and is adjustable to within $\pm$ one timeslot with one bit resolution.

In arbitrary waveform mode, the EVENT 1 connector outputs a timing signal generated by Marker 1.

A marker (3.3 V CMOS high when positive polarity is selected; 3.3 V CMOS low when negative polarity is selected) is output on the EVENT 1 connector whenever a Marker 1 is turned on in the waveform. The damage levels for this connector are $> +8$ V and < -4 V.

12. EVENT 2

This female BNC connector (E8267C PSG only) is used with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this female BNC connector is non-functional.

In real-time mode, the EVENT 2 connector outputs a data enable signal for gating external equipment. This is applicable when external data is clocked into internally generated timeslots. Data is enabled when the signal is low.

In arbitrary waveform mode, the EVENT 2 connector outputs a timing signal generated by Marker 2.

A marker (3.3 V CMOS high when positive polarity is selected; 3.3 V CMOS low when negative polarity is selected) is output on the EVENT 2 connector whenever a Marker 2 is turned on in the waveform. The damage levels for this connector are $> +8$ V and < -4 V.

13. PATTERN TRIG IN

This female BNC connector (E8267C PSG only) is used with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this female BNC connector is non-functional.

This connector accepts a signal that triggers an internal pattern or frame generator to start single pattern output. Minimum pulse width is 100 ns. Damage levels are > +5.5 and < −0.5 V.

14. BURST GATE IN

This female BNC connector (E8267C PSG only) is used with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this female BNC connector is non-functional.

This connector accepts a signal for gating burst power. Burst gating is used when you are externally supplying data and clock information. The input signal must be synchronized with the external data input that will be output during the burst. The burst power envelope and modulated data are internally delayed and re-synchronized. The input signal must be CMOS high for normal burst RF power or CW RF output power and CMOS low for RF off.

Damage levels are > +5.5 and < −0.5 V.

15. AUXILIARY I/O

This female 37-pin connector (E8267C PSG only) is active only on instruments with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this connector is non-functional. This connector provides access to the inputs and outputs described in the following table and shown in Figure 1-5.

Connector Pin	Description
ALT PWR IN	Pin-16 of the Aux I/O connector is used with an internal baseband generator. This pin accepts a CMOS signal for synchronization of external data and alternate power signal timing. Damage levels are > +8 V and < −4 V.
DATA CLK OUT	Pin-6 of the Aux I/O connector is used with an internal baseband generator. This pin relays a CMOS bit clock signal for synchronizing serial data.
DATA OUT	Pin-7 of the Aux I/O connector is used with an internal baseband generator. This pin outputs data (CMOS) from the internal data generator or the externally supplied signal at data input.
EVENT 3	Pin-19 of the Aux I/O connector is used with an internal baseband generator. In arbitrary waveform mode, this pin outputs a timing signal generated by Marker 3. A marker (3.3 V CMOS high when positive polarity is selected; 3.3 V CMOS low when negative polarity is selected) is output on this pin whenever a Marker 3 is turned on in the waveform. The reverse damage levels for this connector are > +8 V and < −4 V.

Connector Pin	Description
EVENT 4	Pin-18 of the Aux I/O connector is used with an internal baseband generator. In arbitrary waveform mode, this pin outputs a timing signal generated by Marker 4. A marker (3.3 V CMOS high when positive polarity is selected; 3.3 V CMOS low when negative polarity is selected) is output on this pin whenever a Marker 4 is turned on in the waveform. The reverse damage levels for this connector are > +8 V and < −4 V.
PATT TRIG IN 2	Pin-17 of the Aux I/O connector accepts a signal that triggers an internal pattern or frame generator to start single pattern output. Minimum pulse width is 100 ns. Damage levels are > +5.5 and < −0.5 V.
SYM SYNC OUT	Pin-5 of the Aux I/O connector is used with an internal baseband generator. This pin outputs the CMOS symbol clock for symbol synchronization, one data clock period wide.

Figure 1-5   **Auxiliary I/O Connector (Female 37-Pin)**
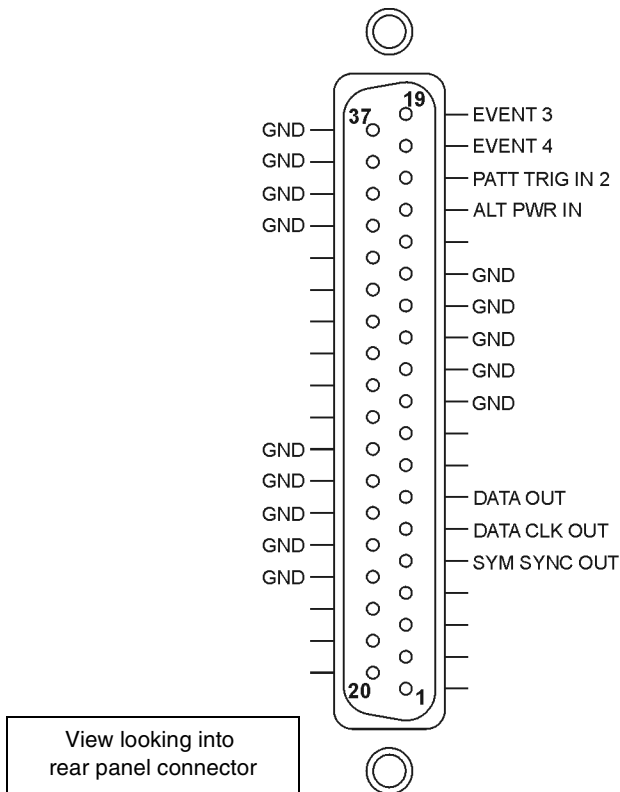
16. Digital Bus

This is a proprietary bus used for Agilent Baseband Studio products, which require the E8267C with Option 602. This connector is not operational for general purpose customer use. Signals are present only when a Baseband Studio option is installed (for details, refer to www.agilent.com/find/basebandstudio).

17. WIDEBAND I INPUT

This female BNC connector (E8267C PSG only) is used with wideband external I/Q inputs (Option 015); on signal generators without Option 015, this female BNC connector is non-functional.

This female BNC connector accepts wide-band AM and allows direct high-bandwidth analog inputs to the I/Q modulator in the 3.2 to 20 GHz frequency range. This input is not calibrated and accepts a 0 dBm maximum power.

18. WIDEBAND Q INPUT

This female BNC connector (E8267C PSG only) is used with wideband external I/Q inputs (Option 015); on signal generators without Option 015, this female BNC connector is non-functional.

This female BNC connector allows direct high-bandwidth analog inputs to the I/Q modulator in the 3.2 to 20 GHz frequency range. This input is not calibrated and accepts a 0 dBm maximum power.

19. COH (COHERENT CARRIER OUTPUT)

This female SMA connector (E8267C PSG only) outputs an RF signal that is phase coherent with the signal generator carrier.

The coherent carrier connector outputs RF that is not modulated with AM, pulse, or I/Q modulation, but is modulated with FM or ΦM (when FM or ΦM are on). The output power is nominally 0 dBm. The output frequency range is from 249.99900001 MHz to 3.2 GHz; this output is not useful for output frequencies > 3.2 GHz.

If the RF output frequency is below 249.99900001 MHz, the coherent carrier output signal will have the following frequency:

- Frequency of coherent carrier = (1E9 − Frequency of RF output) in Hz.

- Damage levels are 20 Vdc and 13 dBm reverse RF power.

20. I OUT

This female BNC connector (E8267C PSG only) can be used with an internal baseband generator (Option 002/602) to output the analog, in-phase component of I/Q modulation; on signal generators without Option 002/602, this female BNC connector can be used to output the in-phase component of an external I/Q modulation that has been fed into the I input connector. The nominal output impedance of the I OUT connector is 50Ω, dc-coupled.

21. I-bar OUT

This female BNC connector (E8267C PSG only) can be used with an internal baseband generator (Option 002/602) to output the complement of the analog, in-phase component of I/Q modulation; on signal generators without Option 002/602, this female BNC connector can be used to output the complement of the in-phase component of an external I/Q modulation that has been fed into the I input connector.

I-bar OUT is used in conjunction with I OUT to provide a balanced baseband stimulus. Balanced signals are signals present in two separate conductors that are symmetrical relative to ground and are opposite in polarity (180 degrees out of phase). The nominal output impedance of the I-bar OUT connector is 50Ω, dc-coupled.

22. Q OUT

This female BNC connector (E8267C PSG only) can be used with an internal baseband generator (Option 002/602) to output the analog, quadrature-phase component of I/Q modulation; on signal generators without Option 002/602, this female BNC connector can be used to output the quadrature-phase component of an external I/Q modulation that has been fed into the Q input connector. The nominal output impedance of the Q OUT connector is 50Ω, dc-coupled.

23. Q-bar OUT

This female BNC connector (E8267C PSG only) can be used with an internal baseband generator (Option 002/602) to output the complement of the analog, quadrature-phase component of I/Q modulation; on signal generators without Option 002/602, this female BNC connector can be used to output the complement of the quadrature-phase component of an external I/Q modulation that has been fed into the Q input connector.

Q-bar OUT is used in conjunction with Q OUT to provide a balanced baseband stimulus. Balanced signals are signals present in two separate conductors that are symmetrical relative to ground and are opposite in polarity (180 degrees out of phase). The nominal output impedance of the Q-bar OUT connector is 50Ω, dc-coupled.

24. BASEBAND GEN REF IN

This female BNC connector (E8267C PSG only) is used with an internal baseband generator (Option 002/602); on signal generators without Option 002/602, this female BNC connector is non-functional.

This connector accepts a 0 to +20 dBm sine wave or TTL square wave signal from an external timebase reference. This external timebase reference clock is used by the internal baseband generator for both component and receiver test applications (only the internal baseband generator can be locked to this external reference; the RF frequency remains locked to the 10 MHz reference).

This connector accepts rates from 250 kHz through 100 MHz; the nominal input impedance is 50Ω at 13 MHz, ac-coupled. The internal clock for the arbitrary waveform generator is locked to this signal when external reference is selected in the ARB setup. The minimum pulse width must be > 10 ns. The damage levels are > +8 V and < −8 V.

25. SMI (SOURCE MODULE INTERFACE)

This interface is used to connect to compatible Agilent Technologies 83550 Series mm-wave source modules.

26. 10 MHz OUT

This female BNC connector outputs a nominal signal level of > +4 dBm and has an output impedance of 50Ω. The accuracy is determined by the timebase being used.

27. 10 MHz IN

This female BNC connector accepts an external timebase reference input signal level of greater than −3 dBm. The reference must be 1, 2, 2.5, 5, or 10 MHz, within ±1 ppm. The signal generator detects when a valid reference signal is present at this connector and automatically switches from internal to external reference operation. The nominal input impedance is 50Ω.

For Option UNR, this BNC connector accepts a signal with a nominal input level of 5 ±5 dBm. The external frequency reference must be 10 MHz, within ±1 ppm. The nominal input impedance is 50Ω with a damage level of ≥ 10 dBm.

28. 10 MHz EFC (Option UNR)

This female BNC input connector accepts an external dc voltage, ranging from −5 to +5 V, for electronic frequency control (EFC) of the internal 10 MHz reference oscillator. This voltage inversely tunes the oscillator about its center frequency approximately −0.0025 ppm/V. The input resistance is greater than 1 MΩ. When not in use, this connector should be shorted using the supplied shorting cap to assure a stable operating frequency.

2   Basic Operation

This chapter describes operations common to all Agilent PSG signal generators and includes the following major sections:

Configuring a Continuous Wave RF Output

This section shows you how to create continuous wave RF output. Using these procedures, you will learn how to perform the following:

- "To Configure an RF Output Frequency" on page 34

- "To Configure an RF Output Frequency Reference and Frequency Offset" on page 35

- "To Configure an RF Output Amplitude" on page 36

- "To Configure an RF Output Amplitude Reference and Amplitude Offset" on page 36

To Configure an RF Output Frequency

The following procedure sets the RF output frequency to 700 MHz and demonstrates how to increment or decrement the output frequency in 1 MHz steps.

1. Press **Preset**.

    This returns the signal generator to the factory-defined state.

NOTE	You can change the preset condition of the signal generator to a user-defined state. For these examples, however, use the factory-defined preset state (the **Preset Normal User** softkey in the Utility menu must be set to Normal).

---

2. Observe the FREQUENCY area of the display (in the upper left-hand corner).

    The value displayed is the maximum specified frequency of the signal generator.

3. Press **RF On/Off**.

    The **RF On/Off** hardkey must be pressed before the RF signal is available at the
    RF OUTPUT connector. The display annunciator changes from RF OFF to RF ON. The maximum specified frequency should be output at the RF OUTPUT connector (at the signal generator's minimum power level).

4. Press **Frequency** > **700** > **MHz**.

    The 700 MHz RF frequency should be displayed in the FREQUENCY area of the display and also in the active entry area.

5. Press **Frequency** > **Incr Set** > **1** > **MHz**.

    This changes the frequency increment value to 1 MHz.

6. Press the up arrow key.

   Each press of the up arrow key increases the frequency by the increment value last set with the **Incr Set** hardkey. The increment value is displayed in the active entry area.

7. The down arrow decreases the frequency by the increment value set in the previous step. Practice stepping the frequency up and down in 1 MHz increments.

   You can also adjust the RF output frequency using the knob. As long as frequency is the active function (the frequency is displayed in the active entry area), the knob will increase and decrease the RF output frequency.

8. Use the knob to adjust the frequency back to 700 MHz.

To Configure an RF Output Frequency Reference and Frequency Offset

The following procedure sets the RF output frequency as a reference frequency to which all other frequency parameters are relative. The frequency initially shown on the display will be 0.00 Hz (the frequency output by the hardware minus the reference frequency). Although the display changes, the frequency output does not change. Any subsequent frequency changes are shown as incremental or decremental to 0 Hz.

1. Press **Preset**.

2. Press **Frequency** > **700** > **MHz**.

3. Press **More (1 of 3)** > **Freq Ref Set**.

   This activates the frequency reference mode and sets the current output frequency (700 MHz) as the reference value. The FREQUENCY area displays 0.000 Hz, which is the frequency output by the hardware (700 MHz) minus the reference value (700 MHz). The REF indicator is activated and the **Freq Ref Off On** softkey has toggled to On.

4. Press **RF On/Off**.

   The display annunciator has changed from RF OFF to RF ON. The RF frequency at the RF OUTPUT connector is 700 MHz.

5. Press **Frequency** > **Incr Set** > **1** > **MHz**.

   This changes the frequency increment value to 1 MHz.

6. Press the up arrow key.

   This increments the output frequency by 1 MHz. The FREQUENCY area display changes to show 1.000 000 000 MHz, which is the frequency output by the hardware (700 MHz + 1 MHz) minus the reference frequency (700 MHz). The frequency at the RF OUTPUT changes to 701 MHz.

7. Press **More (1 of 3)** > **Freq Offset** > **1** > **MHz**.

   This enters a 1 MHz offset. The FREQUENCY area displays 2.000 000 00 MHz, which is the frequency output by the hardware (701 MHz) minus the reference frequency (700 MHz) plus the offset (1 MHz). The OFFS indicator is activated. The frequency at the RF OUTPUT connector is still 701 MHz.

To Configure an RF Output Amplitude

1. Press **Preset**.

2. Observe the AMPLITUDE area of the display.

   The display reads the minimum power level of the signal generator. This is the normal preset RF output amplitude.

3. Press **RF On/Off**.

   The display annunciator changes from RF OFF to RF ON. The RF signal should be output at the minimum power level at the RF OUTPUT connector.

4. Press **Amplitude** > **-20** > **dBm**.

   This changes the amplitude to −20 dBm. The new −20 dBm RF output power should be displayed in the AMPLITUDE area of the display and also in the active entry area.

   Amplitude is still the active function until you press another front panel function key. You can also change the amplitude using the up and down arrow keys and the knob.

To Configure an RF Output Amplitude Reference
and Amplitude Offset

The following procedure sets the RF output power as an amplitude reference to which all other amplitude parameters are relative. The amplitude initially shown on the display will be 0 dB (the power output by the hardware minus the reference power). Although the display changes, the output power does not change. Any subsequent power changes are shown as incremental or decremental to 0 dB.

1. Press **Preset**.

2. Press **Amplitude** > **-20** > **dBm**.

3. Press **More (1 of 2)** > **Ampl Ref Set**.

   This activates the amplitude reference mode and sets the current output power (−20 dBm) as the reference value. The AMPLITUDE area displays 0.00 dB, which is the power output by the hardware (−20 dBm) minus the reference value (−20 dBm). The REF indicator is activated and the **Ampl Ref Off On** softkey has toggled to On.

4. Press **RF On/Off**.

   The display annunciator has changed from RF OFF to RF ON. The power at the RF OUTPUT connector is −20 dBm.

5. Press **Incr Set** > **10** > **dB**.

   This changes the amplitude increment value to 10 dB.

6. Use the up arrow key to increase the output power by 10 dB.

   The AMPLITUDE area displays 10.00 dB, which is the power output by the hardware (-20 dBm plus 10 dBm) minus the reference power (−20 dBm). The power at the RF OUTPUT connector changes to −10 dBm.

7. Press **Ampl Offset** > **10** > **dB**.

   This enters a 10 dB offset. The AMPLITUDE area displays 20.00 dB, which is the power output by the hardware (−10 dBm) minus the reference power (−20 dBm) plus the offset (10 dB). The OFFS indicator is activated. The power at the RF OUTPUT connector is still −10 dBm.

Configuring a Swept RF Output

This section will show you how to create swept RF outputs. Each signal generator has up to three sweep types: step sweep, list sweep, and ramp sweep (Option 007). For ramp sweep, refer to "Using Ramp Sweep (Option 007)" on page 44.

NOTE	List sweep data cannot be saved within an instrument state, but *can* be saved to the memory catalog. For instructions on saving list sweep data, see "Storing Files to the Memory Catalog" on page 67.
	During swept RF output, the FREQUENCY and AMPLITUDE areas of the signal generator's display are deactivated, depending on what is being swept.

This section provides an explanation of the differences between step sweep and list sweep. You will learn two ways to configure the signal generator's RF output to sweep a defined set of frequency and amplitude points. You will create a step sweep and then you will use these points as the basis for a new list sweep.

Using these procedures, you will learn how to perform the following:

Understanding Step Sweep

Step sweep provides a linear progression through the start-to-stop frequency and/or amplitude values. You can toggle the direction of the sweep, up or down. When the **Sweep Direction Down Up** softkey is set to Up, values are swept from the start amplitude/frequency to the stop amplitude/frequency. When set to Down, values are swept from the stop amplitude/frequency to the start amplitude/frequency.

When a step sweep is activated, the signal generator sweeps the RF output based on the values entered for RF output start and stop frequencies and amplitudes, a number of equally spaced points (steps) to dwell upon, and the amount of dwell time at each point; dwell time is the minimum period of time after the settling time that the signal generator will remain at its current state. The frequency, amplitude, or frequency *and* amplitude of the RF output will sweep from the start amplitude/frequency to the stop amplitude/frequency, dwelling at equally spaced intervals defined by the **# Points** softkey value.

To Configure a Step Sweep, in Single Sweep Mode

In this procedure, you will create a step sweep with nine, equally-spaced points, and the following parameters:

- frequency range from 500 MHz to 600 MHz

- amplitude from −20 dBm to 0 dBm

- dwell time 500 ms at each point

1. Press **Preset**.

2. Press **Sweep/List**.

   This opens a menu of sweep softkeys.

3. Press **Sweep Repeat Single Cont**.

   This toggles the sweep repeat from continuous to single.

4. Press **Configure Step Sweep**.

5. Press **Freq Start** > **500** > **MHz**.

   This changes the start frequency of the step sweep to 500 MHz.

6. Press **Freq Stop** > **600** > **MHz**.

   This changes the stop frequency of the step sweep to 600 MHz.

7. Press **Ampl Start** > **-20** > **dBm**.

   This changes the amplitude level for the start of the step sweep.

8. Press **Ampl Stop** > **0** > **dBm**.

    This changes the amplitude level for the end of the step sweep.

9. Press **# Points** > **9** > **Enter**.

    This sets the number of sweep points to nine.

10. Press **Step Dwell** > **500** > **msec**.

    This sets the dwell time at each point to 500 milliseconds.

11. Press **Return** > **Sweep** > **Freq & Ampl**.

    This sets the step sweep to sweep both frequency and amplitude data. Selecting this softkey returns you to the previous menu and turns on the sweep function.

12. Press **RF On/Off**.

    The display annunciator changes from RF OFF to RF ON.

13. Press **Single Sweep**.

    A single sweep of the frequencies and amplitudes configured in the step sweep is executed and available at the RF OUTPUT connector. On the display, the SWEEP annunciator appears for the duration of the sweep and a progress bar shows the progression of the sweep. The **Single Sweep** softkey can also be used to abort a sweep in progress. To see the frequencies sweep again, press Single Sweep to trigger the sweep.

To Configure a Step Sweep, in Continuous Sweep Mode

Press **Sweep Repeat Single Cont**.

This toggles the sweep from single to continuous. A continuous repetition of the frequencies and amplitudes configured in the step sweep are now available at the RF OUTPUT connector. The SWEEP annunciator appears on the display, indicating that the signal generator is sweeping and progression of the sweep is shown by a progress bar.

Understanding List Sweep

List sweep allows you to create a list of arbitrary frequency, amplitude, and dwell time values and sweep the RF output based on the entries in the List Mode Values table.

Unlike a step sweep that contains linear ascending/descending frequency and amplitude values, spaced at equal intervals throughout the sweep, list sweep frequencies and amplitudes can be entered at unequal intervals, nonlinear ascending/descending, or random order.

For convenience, the List Mode Values table can be copied from a previously configured step sweep. Each step sweep point's associated frequency, amplitude and dwell time values are entered into a row in the List Mode Values table, as the following example illustrates.

To Configure a List Sweep,
in Single Sweep Mode, Using Step Sweep Data

In this procedure, you will leverage the step sweep points and change the sweep information by editing several points in the List Mode Values table. For information on using tables, see "Using Tables to Edit Parameters" on page 64.

1. Press **Sweep Repeat Single Cont**.

   This toggles the sweep repeat from continuous to single. The SWEEP annunciator is turned off. The sweep will not occur until it is triggered.

2. Press **Sweep Type List Step**.

   This toggles the sweep type from step to list.

3. Press **Configure List Sweep**.

   This opens another menu displaying softkeys that you will use to create the sweep points. The display shows the current list data. (When no list has been previously created, the default list contains one point set to the signal generator's maximum frequency, minimum amplitude, and a dwell time of 2 ms.)

4. Press **More (1 of 2)** > **Load List From Step Sweep** > **Confirm Load From Step Data**.

   The points you defined in the step sweep are automatically loaded into the list.

To Edit List Sweep Points

1. Press **Return > Sweep > Off**.

   Turning the sweep off allows you to edit the list sweep points without generating errors. If sweep remains on during editing, errors occur whenever one or two point parameters (frequency, power, and dwell) are undefined.

2. Press **Configure List Sweep**.

   This returns you to the sweep list table.

3. Use the arrow keys to highlight the dwell time in row 1.

4. Press **Edit Item**.

   The dwell time for point 1 becomes the active function.

5. Press **100 > msec**.

   This enters 100 ms as the new dwell time value for row 1. Note that the next item in the table (in this case, the frequency value for point 2) becomes highlighted after you press the terminator softkey.

6. Using the arrow keys, highlight the frequency value in row 4.

7. Press **Edit Item > 545 > MHz**.

   This changes the frequency value in row 4 to 545 MHz.

8. Highlight any column in the point 7 row and press **Insert Row**.

   This adds a new point between points 7 and 8. A copy of the point 7 row is placed between points 7 and 8, creating a new point 8, and renumbering the successive points.

9. Highlight the frequency item for point 8, then press **Insert Item**.

   Pressing **Insert Item** shifts frequency values down one row, beginning at point 8. Note that the original frequency values for both points 8 and 9 shift down one row, creating an entry for point 10 that contains only a frequency value (the power and dwell time items do not shift down).

   The frequency for point 8 is still active.

10. Press **590 > MHz**.

11. Press **Insert Item > -2.5 > dBm**.

    This inserts a new power value at point 8 and shifts down the original power values for points 8 and 9 by one row.

12. Highlight the dwell time for point 9, then press **Insert Item**.

    A duplicate of the highlighted dwell time is inserted for point 9, shifting the existing value down to complete the entry for point 10.

To Configure a List Sweep, in Single Sweep Mode

1. Press **Return** > **Sweep** > **Freq & Ampl**

    This turns the sweep on again. No errors should occur if all parameters for every point have been defined in the previous editing process.

2. Press **Single Sweep**.

    The signal generator will single sweep the points in your list. The SWEEP annunciator activates during the sweep.

3. Press **More (1 of 2)** > **Sweep Trigger** > **Trigger Key**.

    This sets the sweep trigger to occur when you press the **Trigger** hardkey.

4. Press **More (2 of 2)** > **Single Sweep**.

    This arms the sweep. The ARMED annunciator is activated.

5. Press the **Trigger** hardkey.

    The signal generator will single sweep the points in your list and the SWEEP annunciator will be activated during the sweep.

To Configure a List Sweep, in Continuous Sweep Mode

Press **Sweep Repeat Single Cont**.

This toggles the sweep from single to continuous. A continuous repetition of the frequencies and amplitudes configured in the list sweep are now available at the RF OUTPUT connector. The SWEEP annunciator appears on the display, indicating that the signal generator is sweeping and progression of the sweep is shown by a progress bar.

Using Ramp Sweep (Option 007)

Ramp sweep provides a linear progression through the start-to-stop frequency and/or amplitude values. Ramp sweep is much faster than step or list sweep and is designed to work with an 8757D scalar network analyzer.

This section describes the ramp sweep capabilities available in PSG signal generators with Option 007. You will learn how to configure the PSG to work with an 8757D scalar network analyzer to perform basic ramp sweep operations. This section comprises the following topics:

- "To Use Basic Ramp Sweep Functions" on page 44

- "To Configure a Ramp Sweep for a Master/Slave Setup" on page 52

- "To Use 8757D Pass-Thru Commands" on page 54

To Use Basic Ramp Sweep Functions

This procedure comprises the tasks listed below. Each task builds upon the previous.

- "Configuring a Frequency Sweep" on page 44

- "Using Markers" on page 47

- "Adjusting Sweep Time" on page 49

- "Using Alternate Sweep" on page 50

- "Configuring an Amplitude Sweep" on page 51

Configuring a Frequency Sweep

1. Set up the equipment as shown in Figure 2-1 on page 45.

NOTE	The PSG signal generator is not compatible with the GPIB system interface in the 8757A, 8757C, or 8757E. For those older scalar network analyzers, do not connect the GPIB cable in Figure 2-1 on page 45. This method provides only a subset of 8757D functionality. See the *PSG Data Sheet* for details. Use the 8757A/C/E documentation instead of this procedure.

Figure 2-1　　　　Equipment Setup
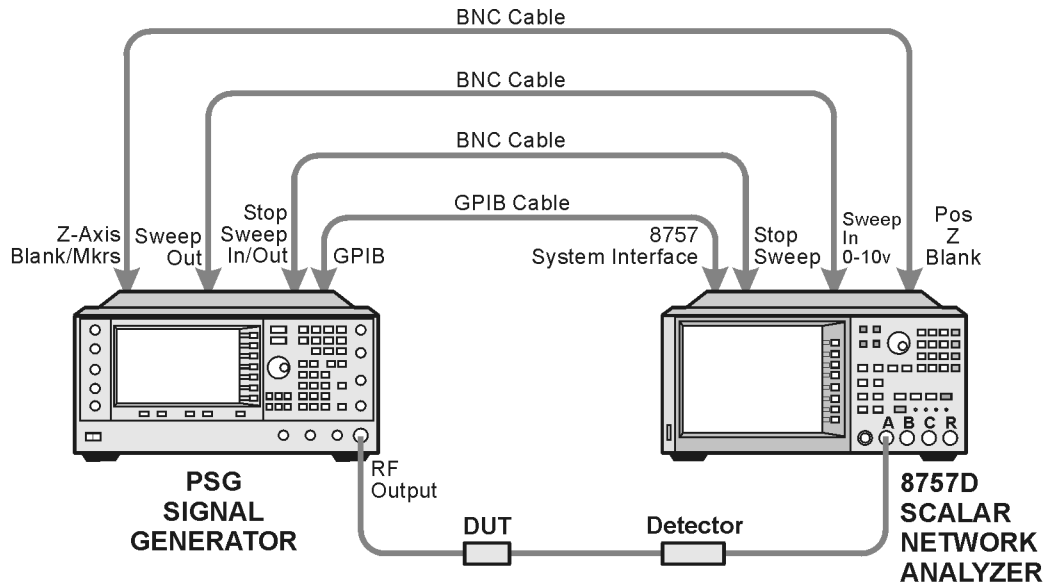
scaler_netwk_1

2. Turn on both the 8757D and the PSG.

3. On the 8757D, press **SYSTEM** > **MORE** > **SWEEP MODE** > and verify that the **SYSINTF** softkey is set to ON.

   This ensures that the system interface mode is activated on the 8757D. The system interface mode allows the instruments to work together as a system.

4. Press **Utility** > **GPIB/RS-232 LAN** to view the PSG's GPIB address under the **GPIB Address** softkey. If you want to change it, press **GPIB Address** and use the front panel knob or numeric key pad to change the value.

5. On the 8757D, press **LOCAL** > **SWEEPER** to verify that the GPIB address matches that of the PSG. If it doesn't match, use the numeric keypad and press **ENT** to change the value.

6. Preset either instrument.

   Presetting one of the instruments should automatically preset the other as well. If both instruments do not preset, check the GPIB connection, GPIB addresses, and ensure the 8757D is set to system interface mode (**SYSINTF** set to ON).

   The PSG automatically activates a 2 GHz to maximum frequency ramp sweep with a constant amplitude of 0 dBm. Notice that the RF ON, SWEEP, and PULSE annunciators appear on the PSG display. The PULSE annunciator appears because the 8757D is operating in AC mode.

   The PSG also switches its remote language setting to *8757D System*, allowing the PSG to talk to the 8757D during ramp sweep operations. You can confirm this by pressing **Utility > GPIB/RS-232 LAN** and observing the selection under the **Remote Language** softkey.

NOTE	During swept RF output, the FREQUENCY and/or AMPLITUDE areas of the signal generator's display are deactivated, depending on what is being swept. In this case, since frequency is being swept, nothing appears in the FREQUENCY area of the display.

7. Press **Frequency > Freq CW**.

   The current continuous wave frequency setting now controls the RF output and ramp sweep is turned off.

8. Press **Freq Start**.

   The ramp sweep settings once again control the RF output and the CW mode is turned off. Pressing any one of the softkeys **Freq Start**, **Freq Stop**, **Freq Center**, or **Freq Span** activates a ramp sweep with the current settings.

NOTE	In a frequency ramp sweep, the start frequency must be lower than the stop frequency.

9. Adjust the settings for **Freq Center** and **Freq Span** so that the frequency response of the device under test (DUT) is clearly seen on the 8757D display.

   Notice how adjusting these settings also changes the settings for the **Freq Start** and **Freq Stop** softkeys. You may need to rescale the response on the 8757D for a more accurate evaluation of the amplitude. Figure 2-2 on page 47 shows an example of a bandpass filter response.

Figure 2-2   **Bandpass Filter Response on 8757D**


```
CH1:  A
 5.0 dB/     REF   −40.00 dBm
```

```
1    >
```

```
1
```

```
START  9.20000  GHz                    STOP  10.2000  GHz
```
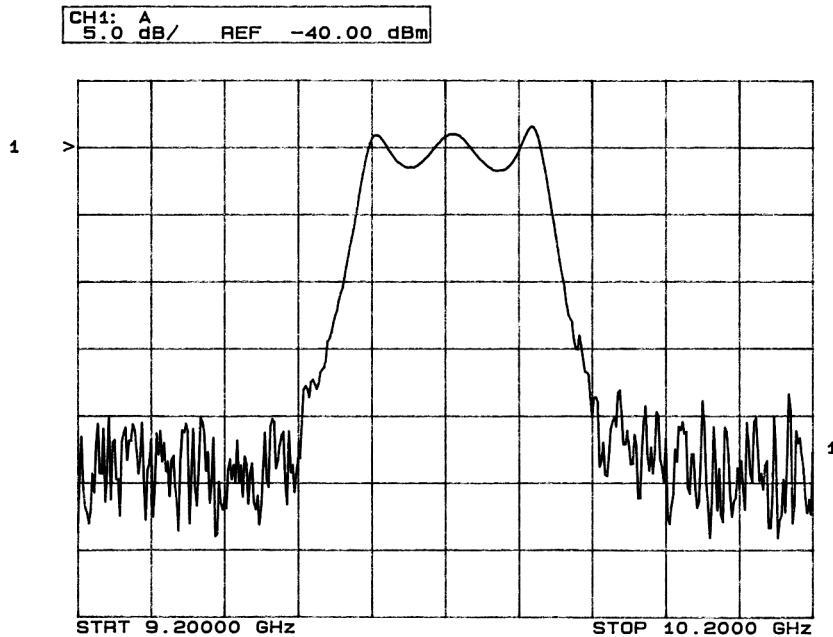
Using Markers

1. Press **Markers**.

   This opens a table editor and associated marker control softkeys. You can use up to 10 different markers, labeled 0 through 9.

2. Press **Marker Freq** and select a frequency value within the range of your sweep.

   In the table editor, notice how the state for marker 0 automatically turns on. The marker also appears on the 8757D display.

3. Use the arrow keys to move the cursor in the table editor to marker 1 and select a frequency value within the range of your sweep, but different from marker 0.

   Notice that marker 1 is activated and is the currently selected marker, indicated by the marker arrow pointing down. As you switch between markers, using the arrow keys, you will notice that the selected marker's arrow points down, while all others point up.

   Notice also that the frequency and amplitude data for the currently selected marker is displayed on the 8757D.

4. Move the cursor back to marker 0 and press **Delta Ref Set** > **Marker Delta Off On** to On.

   In the table editor, notice that the frequency values for each marker are now relative to marker 0. Ref appears in the far right column (also labeled Ref) to indicate which marker is the reference. Refer to Figure 2-3.

   Figure 2-3   Marker Table Editor

5. Move the cursor back to marker 1 and press **Marker Freq**. Turn the front panel knob while observing marker 1 on the 8757D.

   On the 8757D, notice that the displayed amplitude and frequency values for marker 1 are relative to marker 0 as the marker moves along the trace. Refer to Figure 2-4.

Figure 2-4   **Delta Markers on 8757D**
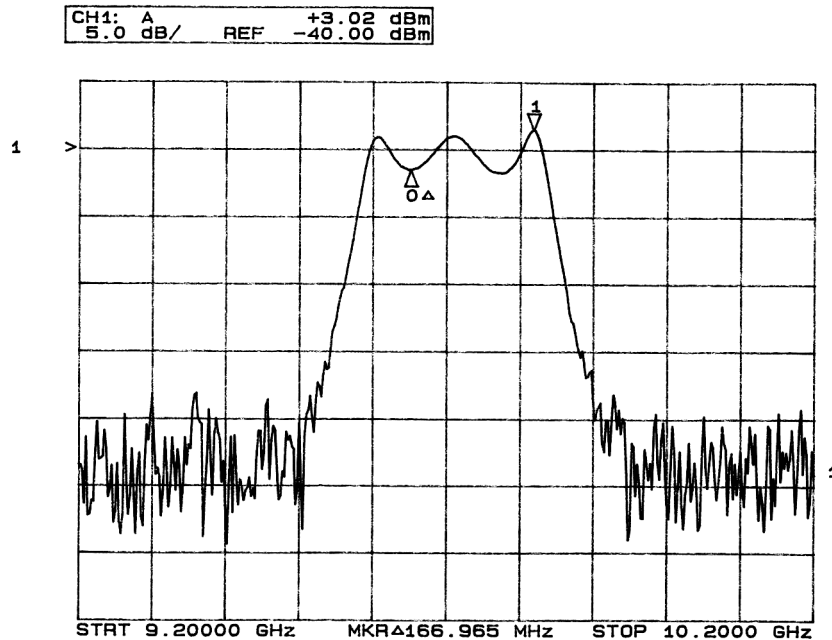
6. Press **Turn Off Markers**.

   All active markers turn off. Refer to the *Key Reference* for information on other marker softkey
   functions.

Adjusting Sweep Time

1. Press **Sweep/List**.

   This opens a menu of sweep control softkeys and displays a status screen summarizing all the current
   sweep settings.

2. Press **Configure Ramp/Step Sweep**.

   Since ramp is the current sweep type, softkeys in this menu specifically control ramp sweep settings.
   When step is the selected sweep type, the softkeys control step sweep settings. Notice that the **Freq Start**
   and **Freq Stop** softkeys appear in this menu in addition to the **Frequency** hardkey menu.

3. Press **Sweep Time** to Manual > **5** > **sec**.

   In auto mode, the sweep time automatically sets to the fastest allowable value. In manual mode, you can select any sweep time slower than the fastest allowable. The fastest allowable sweep time is dependent on the number of trace points and channels being used on the 8757D and the frequency span.

4. Press **Sweep Time** to Auto.

   The sweep time returns to its fastest allowable setting.

Using Alternate Sweep

1. Press the **Save** hardkey.

   This opens the table editor and softkey menu for saving instrument states. Notice that the **Select Reg** softkey is active. (For more information on saving instrument states refer to "To Use the Instrument State Register" on page 67.)

2. Turn the front panel knob until you find an *available* register and press **SAVE**. Remember this saved register number. If no registers are available, you can write over an *in-use* register, by pressing **Re-SAVE**.

NOTE	When you are using the PSG in a system with an 8757 network analyzer, you are limited to using registers 1 through 9 in sequence 0 for saving and recalling states.

---

3. Press **Sweep/List** > **Configure Ramp/Step Sweep** and enter new start and stop frequency values for the ramp sweep.

4. Press **Alternate Sweep Register** and turn the front panel knob to select the register number of the previously saved sweep state.

5. Press **Alternate Sweep Off On** to On.

   The signal generator alternates between the original saved sweep and the current sweep. You may need to adjust 8757D settings to effectively view both sweeps, such as setting channel 2 to measure sensor A. Refer to Figure 2-5.

Figure 2-5 **Alternating Sweeps on 8757D**


```
CH1:  A                              CH2:  A
10.0  dB/    REF    -30.00 dBm      10.0  dB/    REF    -30.00 dBm
```

```
STRT  9.20000  GHz                  STOP  10.2000  GHz
STRT  9.50000  GHz                  STOP  20.0000  GHz
```
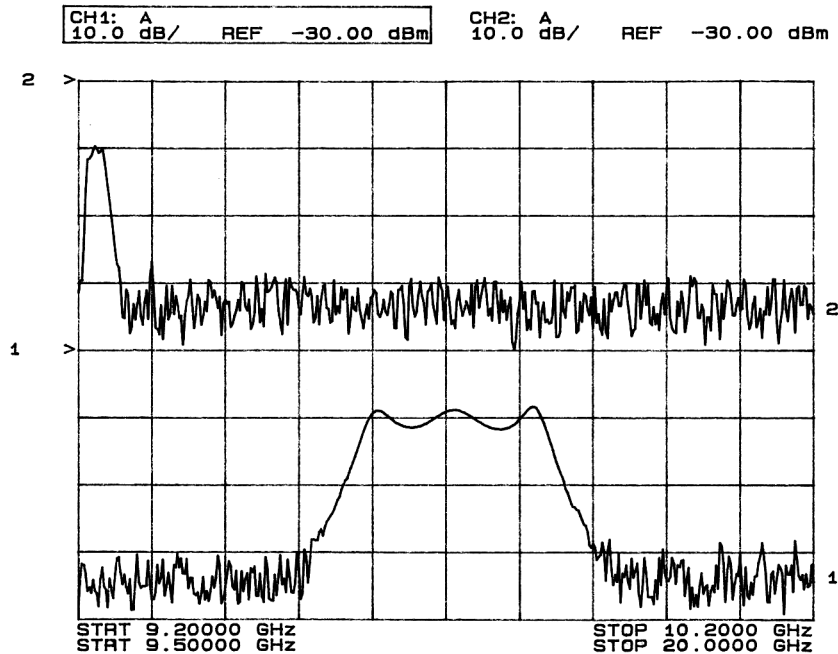
Configuring an Amplitude Sweep

1. Press **Return** > **Sweep** > **Off**.

    This turns off both the current sweep and the alternate sweep from the previous task. The current CW settings now control the RF output.

2. Press **Configure Ramp/Step Sweep**.

3. Using the **Ampl Start** and **Ampl Stop** softkeys, set an amplitude range to be swept.

4. Press **Return** > **Sweep** > **Ampl**.

    The new amplitude ramp sweep settings control the RF output and the CW mode is turned off.

To Configure a Ramp Sweep for a Master/Slave Setup

This procedure shows you how to configure two PSGs and an 8757D to work in a master/slave setup.

1. Set up the equipment as shown in Figure 2-6. Use a 9-pin, D-subminiature, male RS-232 cable with the pin configuration shown in Figure 2-7 on page 53 to connect the auxiliary interfaces of the two PSGs. You can also order the cable (part number 8120-8806) from Agilent Technologies.

   By connecting the master PSG's 10 MHz reference standard to the slave PSG's 10 MHz reference input, the master's timebase supplies the frequency reference for both PSGs.

Figure 2-6   Master/Slave Equipment Setup
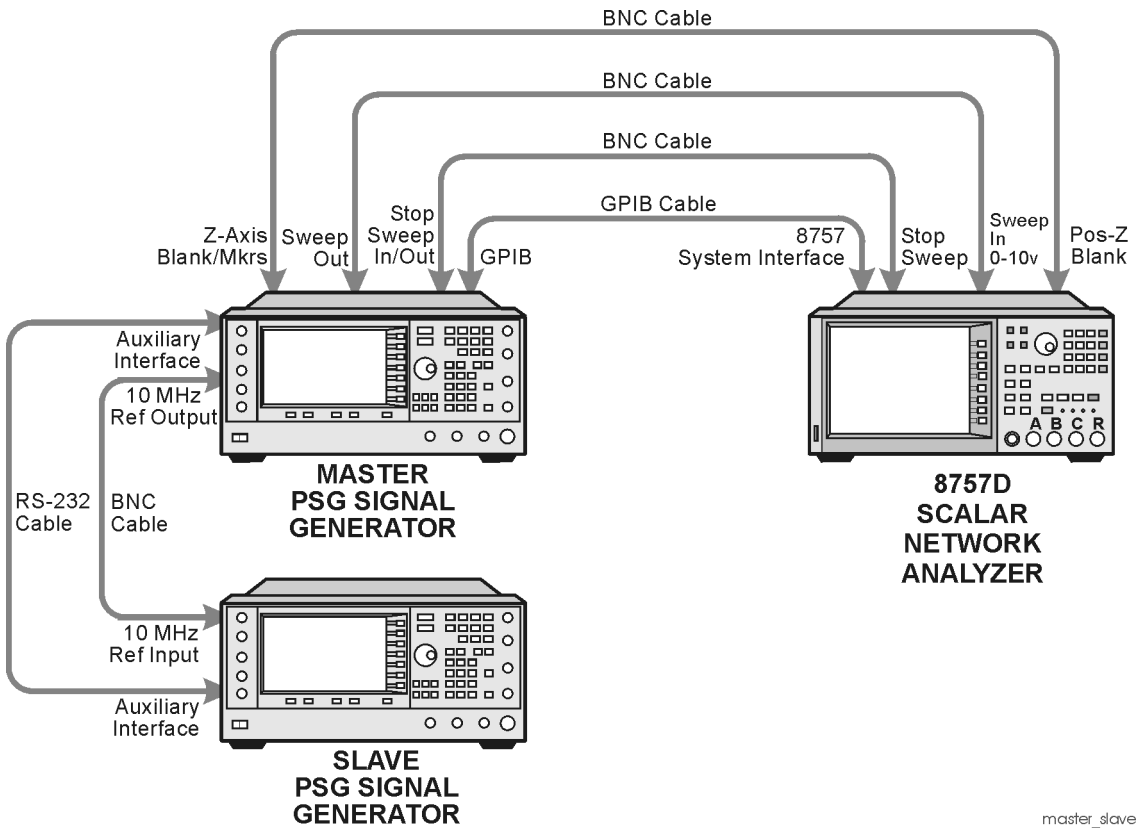
Figure 2-7   **RS-232 Pin Configuration**

Pin 1 ———————— Retrace ———————— Pin 1

Pin 4 ———————— Sweep Stop ———————— Pin 4

Pin 5 ———————— Ground ———————— Pin 5

Connector Type: D-Subminiature, male
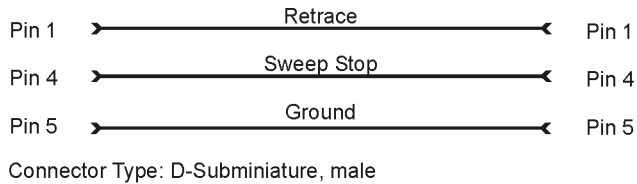
pins145

2. Set up the slave PSG's frequency and power settings.

    By setting up the slave first, you avoid synchronization problems.

3. Set up the master PSG's frequency, power, and sweep time settings.

    The two PSGs can have different frequency and power settings for ramp sweep.

4. Set the slave PSG's sweep time to match that of the master.

    Sweep times must be the same for both PSGs.

5. Set the slave PSG to continuous triggering.

    The slave must be set to continuous triggering, but the master can be set to any triggering mode.

6. On the slave PSG, press **Sweep/List** > **Sweep Type** > **Sweep Control** > **Slave**.

    This sets the PSG to operate in slave mode.

7. On the master PSG, press **Sweep/List** > **Sweep Type** > **Sweep Control** > **Master**.

    This sets the PSG to operate in master mode.

To Use 8757D Pass-Thru Commands

Pass-thru commands allow you to temporarily interrupt ramp sweep system interaction so that you can send operating instructions to the PSG. This section provides setup information and an example program for using pass-thru commands in a ramp sweep system.

Equipment Setup

To send pass-thru commands, set up the equipment as shown in Figure 2-8. Notice that the GPIB cable from the computer is connected to the GPIB interface bus of the 8757D.

Figure 2-8
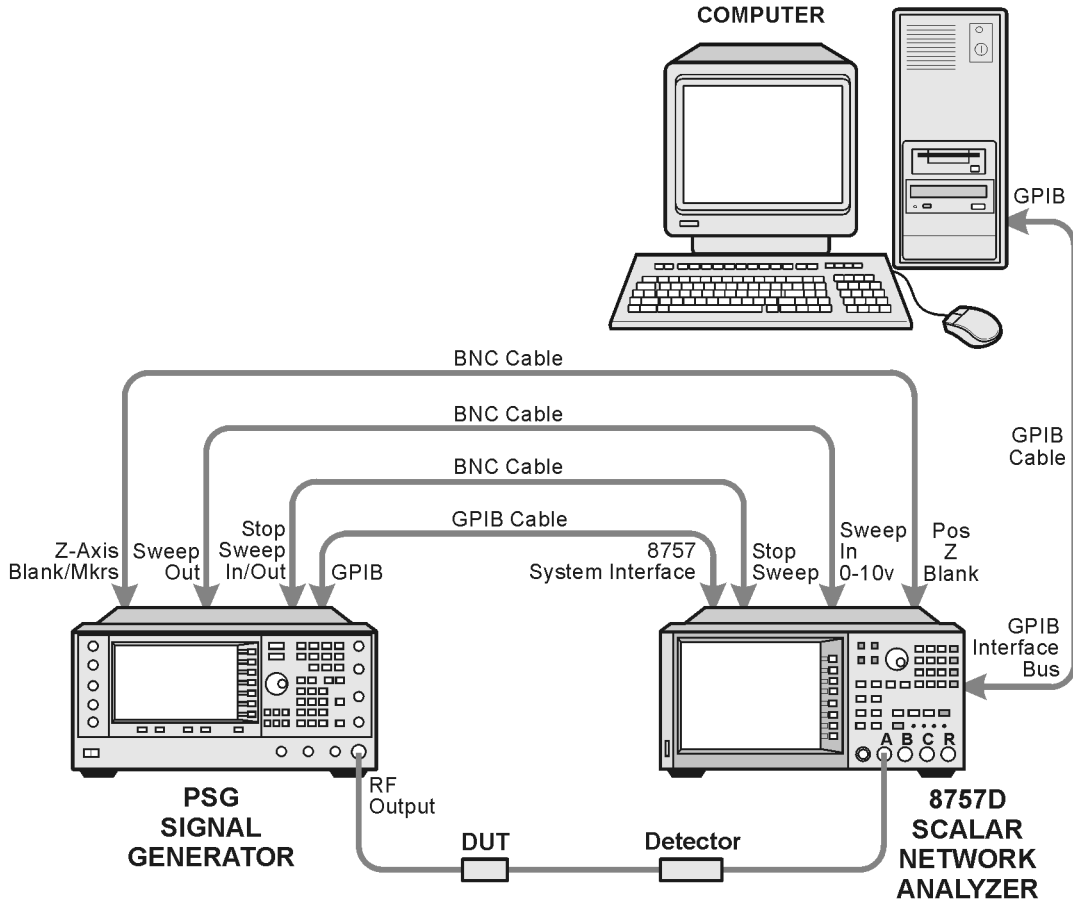
scaler_netwk_pc

GPIB Address Assignments

Table 2-1 describes how GPIB addresses should be assigned for sending pass-thru commands. These are the same addresses used in Example 2-1.

Table 2-1

Instrument	GPIB Address	Key Presses/Description
PSG	19	Press **Utility** > **GPIB/RS-232 LAN** > **GPIB Address** > **19** > **Enter**.
8757D	16	Press **LOCAL** > **8757** > **16** > **Enter**.
8757D (Sweeper)	19	This address must match the PSG. Press **LOCAL** > **SWEEPER** > **19** > **Enter**.
Pass Thru	17	The pass thru address is automatically selected by the 8757D by inverting the last bit of the 8757D address. Refer to the 8757D documentation for more information. Verify that no other instrument is using this address on the GPIB bus.

Example Pass-Thru Program

Example 2-1 on page 56 is a sample Agilent BASIC program that switches the 8757D to pass-thru mode, allowing you to send operating commands to the PSG. After the program runs, control is given back to the network analyzer. The following describes the command lines used in the program.

Line 30	PT is set to equal the source address. C1 is added, but not needed, to specify the channel.
Lines 40, 90	The END statement is required to complete the language transition.
Lines 50, 100	A WAIT statement is recommended after a language change to allow all instrument changes to be completed before the next command.
Lines 70, 80	This is added to ensure that the instrument has completed all operations before switching languages.
Line 110	This takes the network analyzer out of pass-thru command mode, and puts it back in control. Any analyzer command can now be entered.

Example 2-1   Pass-Thru Program

```
10 ABORT 7
20 CLEAR 716
30 OUTPUT 716;"PT19;C1"
40 OUTPUT 717;"SYST:LANG SCPI";END
50 WAIT .5
60 OUTPUT 717;"POW:STAT OFF"
70 OUTPUT 717;"*OPC?"
80 ENTER 717; Reply
90 OUTPUT 717;"SYST:LANG COMP";END
100 WAIT .5
110 OUTPUT 716;"C2"
120 END
```

Extending the Frequency Range with a mm-Wave Source Module

The RF output frequency of the signal generator can be multiplied using an Agilent 83550 Series millimeter-wave source module. The signal generator/mm-wave source module's output is *automatically leveled* when the instruments are connected. The output frequency range depends on the specific mm-wave source module.

NOTE	To ensure adequate RF amplitude at the mm-wave source module RF input, when using an E8267C PSG, E8247C PSG with Option 1EA, or E8257C PSG with Option 1EA, maximum amplitude loss through the adapters and cables connected between the signal generator's RF output and the mm-wave source module's RF input should be less than 1.5 dB.

Required Equipment

- Agilent 83550 Series millimeter-wave source module

- Agilent 8349B microwave amplifier

  Signal generators without Option 1EA (E8247C PSG and E8257C PSG) require an Agilent 8349B microwave amplifier. Signal generators with Option 1EA can drive the output of millimeter-wave source modules to maximum specified power without a microwave amplifier.

- cables and adapters as required

Connect the Equipment

CAUTION	To prevent damage to the signal generator, turn off the line power to the signal generator before connecting the source module interface cable to the rear panel SOURCE MODULE interface connector.

1. Turn off the signal generator's line power.

2. Connect the equipment as shown.

   - E8247C PSG and E8257C PSG *without* Option 1EA, use the setup in Figure 2-9.

   - E8267C PSG or E8247C PSG and E8257C PSG with Option 1EA, use the setup in Figure 2-10.

Figure 2-9   **Setup for E8247C PSG and E8257C PSG *without* Option 1EA**
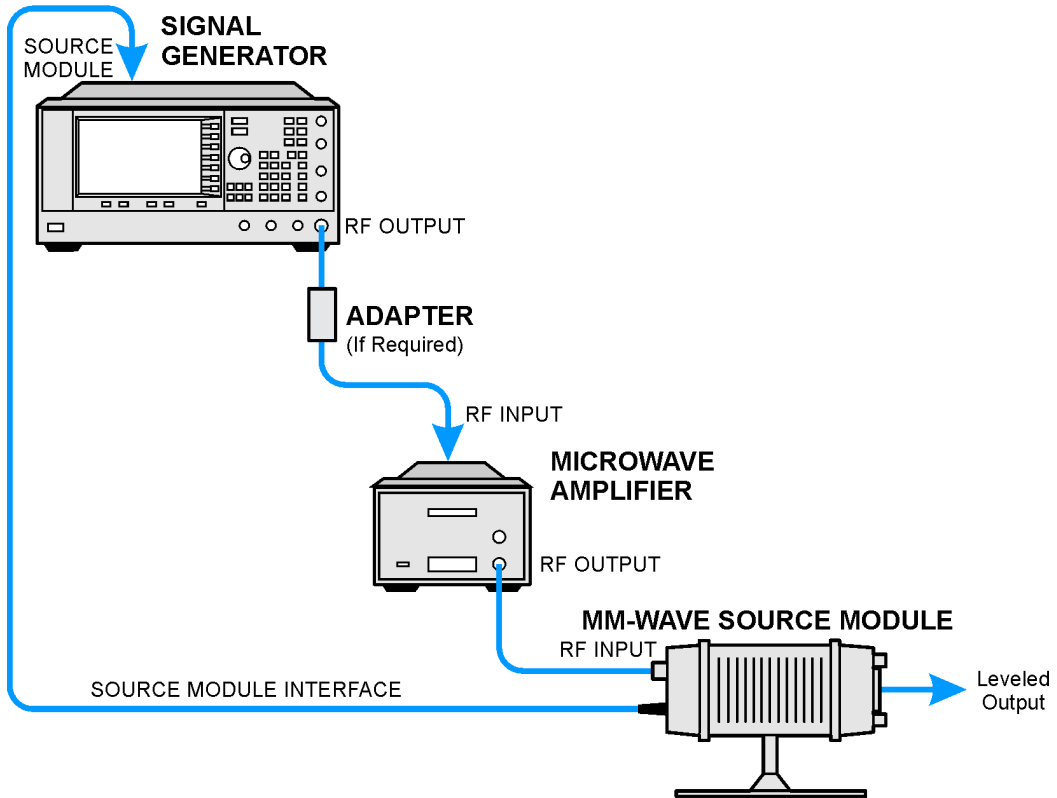
Figure 2-10   Setup for E8267C PSG or E8247C PSG and E8257C PSG with Option 1EA
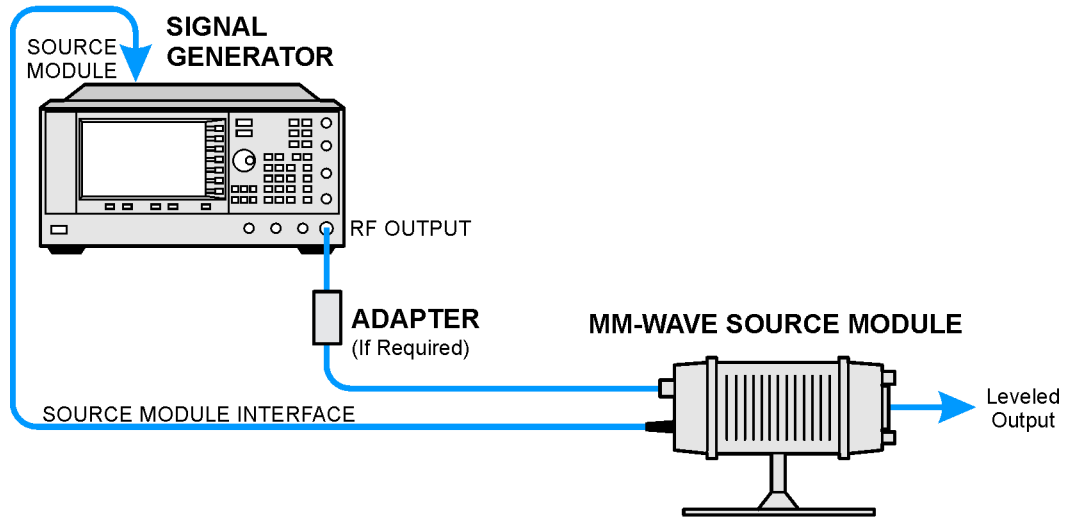
To Configure the Signal Generator

1. Turn on the signal generator's line power.

    Upon power-up, the signal generator automatically:

    • senses the mm-wave source module,

    • switches the signal generator's leveling mode to external/source module (power is leveled at the mm-wave source module output),

    • sets the mm-wave source module frequency and amplitude to the source module's preset values, and

    • in the FREQUENCY and AMPLITUDE areas of the signal generator, displays the RF output frequency and amplitude values available at the mm-wave source module output.

    The MMMOD indicator in the FREQUENCY area and the MM indicator in the AMPLITUDE area of the signal generator's display indicate that the mm-wave source module is active.

NOTE   Refer to the mm-wave source module specifications for the specific frequency and amplitude ranges.

---

2. If the RF OFF annunciator is displayed, press **RF On/Off**.

Leveled power should be available at the output of the millimeter-wave source module.

To obtain flatness-corrected power, refer to "Creating and Applying User Flatness Correction" on page 78.

Turning On Modulation Formats

A modulation format can be turned on prior to or after setting your signal parameters.

To Turn a Modulation Format On

1. Access the first menu within the modulation format.

   This menu will show a softkey that has the format's name associated with off and on. For example, **AM >
   AM Off On**. For some formats, the off/on key may appear in additional menus other than the first one.

2. Press the modulation format off/on key until On is highlighted.

   Figure 2-11 shows the AM modulation format's first menu with off as the format status, and Figure 2-12
   shows an example of the PSG display when the format is active.

The modulation format should be generated, however the carrier signal is still not modulated until the **Mod
On/Off** key has been set to On.

Depending on the modulation format, the signal generator may require a few seconds to build the signal.
Within the digital formats (E8267C PSG with Option 002/602 only), you may see a BaseBand
Reconfiguring status bar appear on the display. Once the signal is generated, an annunciator showing the
name of the format will appear on the display indicating that the modulation format is active. For digital
formats (E8267C PSG with Option 002/602 only), the I/Q annunciator will appear in addition to the name
of the modulation format.

Figure 2-11   Example of AM Modulation Format Off

First AM Menu
Modulation Format is Off

Figure 2-12   **Modulation Format On**

Active Modulation Format Annunciator

First AM Menu
Modulation format is On

Applying Modulation Formats to the RF Output

The carrier signal is modulated when the **Mod On/Off** key is set to On and an individual modulation format is active. When the key is set to On, the MOD ON annunciator shows in the display. The MOD OFF annunciator appears when the key is set to Off. The MOD ON annunciator may be showing even when there are no active modulation formats; this merely indicates that the carrier signal will be modulated when a modulation format is turned on.

To Turn RF Output Modulation On

Press the **Mod On/Off** key until the MOD ON annunciator appears in the display.

The carrier signal should be modulated with all active modulation formats. This is the factory default.

To Turn RF Output Modulation Off

Press the **Mod On/Off** key until the MOD OFF annunciator appears in the display.

The carrier signal is no longer modulated or capable of being modulated when a modulation format is active.

Figure 2-13   Carrier Signal Modulation Status

Mod Set to On—Carrier is Modulated

AM Modulation Format is Active

Mod Set to Off—Carrier is not Modulated

AM Modulation Format is Active

Mod Set to On—Carrier is not Modulated

No Active Modulation Format
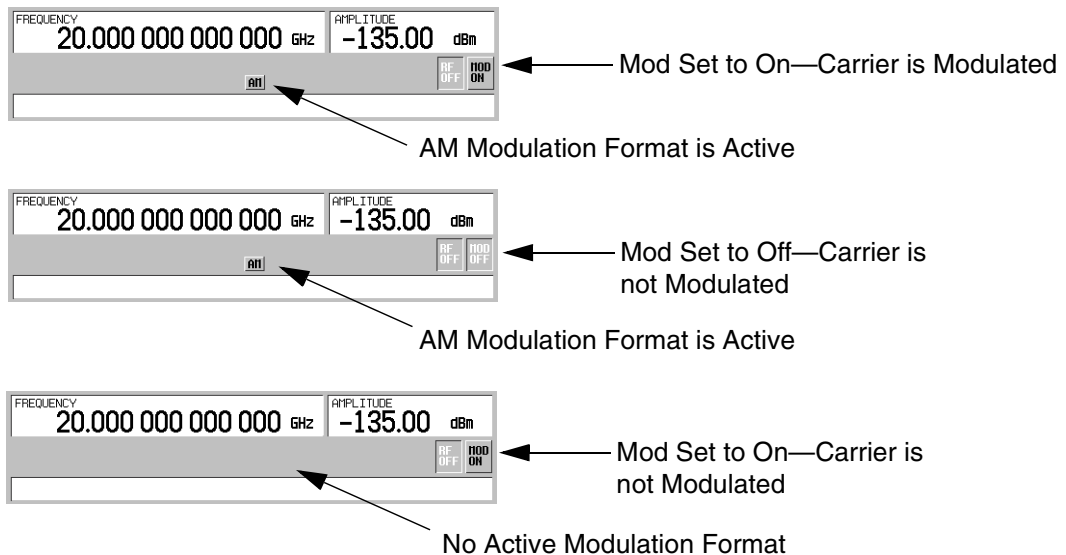
Using Tables to Edit Parameters

Tables enable you to perform configuration tasks such as: create a list sweep, modify the Memory Catalog, modify existing parameters for modulations, as well as others.

The following example shows a table of parameters being edited for List Mode.

Figure 2-14

| Active Function Area | | Cursor |

| Table Name | Table Items | Table Softkeys |
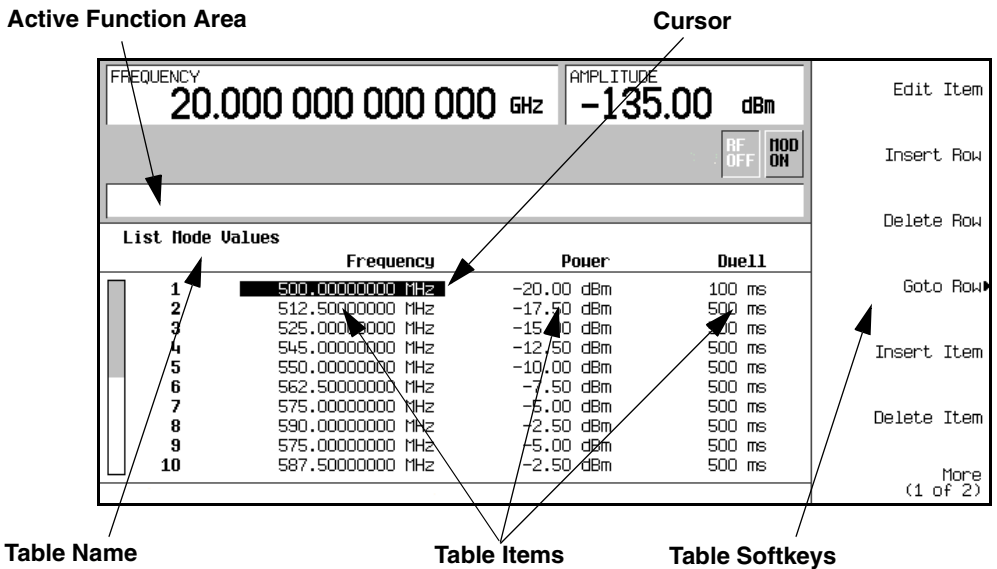
Active Function Area	displays the active table item while its value is edited
Cursor	an inverse video identifier used to highlight specific table items for selection and editing
Table Softkeys	select table items, preset table values, and modify table structures
Table Items	values arranged in numbered rows and titled columns (The columns are also known as data fields. For example, the column below the Frequency title is known as the Frequency data field).

Table Softkeys

The following table softkeys are used to load, navigate, modify, and store table item values.

Edit Item	displays the selected item in the active function area of the display where the item's value can be modified
Insert Row	inserts an identical row of table items above the currently selected row
Delete Row	deletes the currently selected row
Goto Row	opens a menu of softkeys (**Enter**, **Goto Top Row**, **Goto Middle Row**, **Goto Bottom Row**, **Page Up**, and **Page Down**) used to quickly navigate through the table items
Insert Item	inserts an identical item in a new row below the currently selected item
Delete Item	deletes the item from the bottom row of the currently selected column
Page Up and **Page Down**	displays table items that occupy rows outside the limits of the ten-row table display area
More (1 of 2)	accesses **Load/Store** and its associated softkeys
Load/Store	opens a menu of softkeys (**Load From Selected File**, **Store To File**, **Delete File**, **Goto Row**, **Page Up**, and **Page Down**) used to load table items from a file in the memory catalog, or to store the current table items as a file in the memory catalog

To Modify Existing Table Items in the Data Fields

1. Press **Preset** > **Sweep/List** > **Configure List Sweep**.

   The signal generator displays the List Mode Values table, as shown.

2. Use the arrow keys or the knob to move the table cursor over the desired item.

   In Figure 2-14 on page 64, the first item in the Frequency data field has been selected.

3. Press **Edit Item**.

   The selected item is displayed in the active function area of the display.

4. Use the knob, arrow keys, or the numeric keypad to modify the value.

5. Press **Enter**.

   The modified item should be displayed in the table.

Using Data Storage Functions

This section explains how to use the two forms of signal generator data storage: the memory catalog and the instrument state register.

To Use the Memory Catalog

The Memory Catalog is the signal generator's interface for viewing, storing, and saving files; it can be accessed through the signal generator's front panel or a remote controller. (For information on performing these tasks remotely, see the Programming Guide.)

Table 2-2   **Memory Catalog File Types and Associated Data**

Binary	binary data
State	instrument state data (controlling instrument operating parameters, such as frequency, amplitude, and mode)
LIST	sweep data from the List Mode Values table including frequency, amplitude, and dwell time
User Flatness	user flatness calibration correction pair data (user-defined frequency and corresponding amplitude correction values)
FIR	Finite Impulse Response (FIR) filter coefficients
ARB Catalog Types	(E8267C PSG with Option 002/602 only) user created files - Waveform Catalog Types: WFM1 (waveform file), NVARB Catalog Types: NVWFM (non-volatile, ARB waveform file), NVMKR (non-volatile, ARB waveform marker file), Seq (ARB sequence file), MTONE (ARB multitone file), DMOD (ARB digital modulation file), MDMOD (ARB multicarrier digital modulation file)
Modulation Catalog Types	(E8267C PSG with Option 002/602 only) associated data for I/Q and FSK (frequency shift keying) modulation files
Shape	burst shape of a pulse
Bit	Bit

Storing Files to the Memory Catalog

To store a file to the memory catalog, first create a file. For this example, use the default list sweep table.

1. Press **Preset**.

2. Press **Sweep/List** > **Configure List Sweep** > **More (1 of 2)** > **Load/Store**.

   This opens the "Catalog of List Files".

3. Press **Store to File**.

   This displays a menu of alphabetical softkeys for naming the file. Store to: is displayed in the active function area.

4. Enter the file name LIST1 using the alphabetical softkeys and the numeric keypad (for the numbers 0 to 9).

5. Press **Enter**.

   The file should be displayed in the "Catalog of List Files", showing the file name, file type, file size, and the date and time the file was modified.

Viewing Stored Files in the Memory Catalog

1. Press **Utility** > **Memory Catalog** > **Catalog Type**.

   All files in the memory catalog are listed in alphabetical order, regardless of which catalog type you select. File information appears on the display and includes the file name, file type, file size, and the date and time the file was modified.

2. Press **List**.

   The "Catalog of List Files" is displayed.

3. Press **Catalog Type** > **State**.

   The "Catalog of State Files" is displayed.

4. Press **Catalog Type** > **All**.

   The "Catalog of All Files" is displayed. For a complete list of file types, refer to Table 2-2 on page 66.

To Use the Instrument State Register

The instrument state register is a section of memory divided into 10 sequences (numbered 0 through 9) each containing 100 registers (numbered 00 through 99). It is used to store and recall instrument settings. It provides a quick way to reconfigure the signal generator when switching between different signal configurations. Once an instrument state has been saved, you can recall the instrument settings for that state with minimum effort.

NOTE	List sweep data is not saved within an instrument state. For instructions on saving list sweep data, see "Storing Files to the Memory Catalog" on page 67.

Saving an Instrument State

Using this procedure, you will learn how to save current signal generator settings to the instrument state register.

1. Press **Preset**.

2. Configure the signal generator with the following settings:

   a. Press **Frequency** > **800** > **MHz**.

   b. Press **Amplitude** > **0** > **dBm**.

   c. Press **AM** > **AM Off On**.

      This enables amplitude modulation (the AM annunciator should be on).

3. Press **Save** > **Select Seq**.

   The sequence number becomes the active function. The signal generator displays the last sequence that you have used. Set the sequence to 1 using the arrow keys.

4. Press **Select Reg**.

   The register number in sequence 1 becomes the active function. The signal generator displays either the last register used, accompanied by the text: (in use), or (if no registers are in use) register 00, accompanied by the text: (available). Use the arrow keys to select register 01.

5. Press **Save Seq[1] Reg[01]**.

   This will save this instrument state in sequence 1, register 01 of the instrument state register.

6. Press **Add Comment to Seq[1] Reg[01]**.

   This allows you to add a descriptive comment to sequence 1 register 01.

7. Enter your comment using the alphanumeric softkeys or the knob and press **Enter**.

8. Press **Edit Comment In Seq[1] Reg[01]**.

   This allows you to change the descriptive comment for sequence 1 register 01, if desired. Change your comment using the alphanumeric softkeys and press **Enter**.

After making changes to an instrument state, you may save it back to a specific register by highlighting that register and pressing **Re-SAVE Seq[n] Reg[nn]**.

Recalling an Instrument State

Using this procedure, you will learn how to recall instrument settings saved to an instrument state register.

1. Press **Preset**.

2. Press the **Recall** hardkey.

   Notice that the **Select Seq** softkey shows sequence 1. (This is the last sequence that you used.)

3. Press **RECALL Reg**.

   The register to be recalled in sequence 1 becomes the active function. Press the up arrow key once to select register 1. Your stored instrument state settings should have been recalled.

Deleting Registers and Sequences

Using this procedure, you will learn how to delete registers and sequences saved to an instrument state register.

To Delete a Specific Register within a Sequence

1. Press **Preset**.

2. Press the **Recall** or **Save** hardkey.

   Notice that the **Select Seq** softkey shows the last sequence that you used.

3. Press **Select Seq** and enter the sequence number containing the register you want to delete.

4. Press **Select Reg** and enter the register number you want to delete.

   Notice that the **Delete Seq[n] Reg[nn]** should be loaded with the sequence and register you want to delete.

5. Press **Delete Seq[n] Reg[nn]**.

   This deletes the chosen register.

To Delete All Registers within a Sequence

1. Press **Preset**.

2. Press the **Recall** or **Save** hardkey.

   Notice that the **Select Seq** softkey shows the last sequence that you used.

3. Press **Select Seq** and enter the sequence number containing the registers you want to delete.

4. Press **Delete all Regs in Seq[n]**.

   This deletes all registers in the selected sequence.

To Delete All Sequences

|	
CAUTION	This will delete the contents of all registers and all sequences contained in the instrument state register.

1. Press **Preset**.

2. Press the **Recall** or **Save** hardkey.

   Notice that the **Select Seq** softkey shows the last sequence that you used.

3. Press **Delete All Sequences**.

   This deletes all of the sequences saved in the instrument state register.

Enabling Options

You can retrofit your signal generator after purchase to add new capabilities. Some new optional features are implemented in hardware that you must install. Some options are implemented in software, but require the presence of optional hardware in the instrument. This example shows you how to enable software options.

To Enable a Software Option

1. A license key is required to enable each software option. This license key is provided on the license key certificate that you receive when you purchase the software option. Access the Software Options menu by pressing **Utility** > **Instrument Adjustments** > **Instrument Options** > **Software Options**. An example of the signal generator display follows:
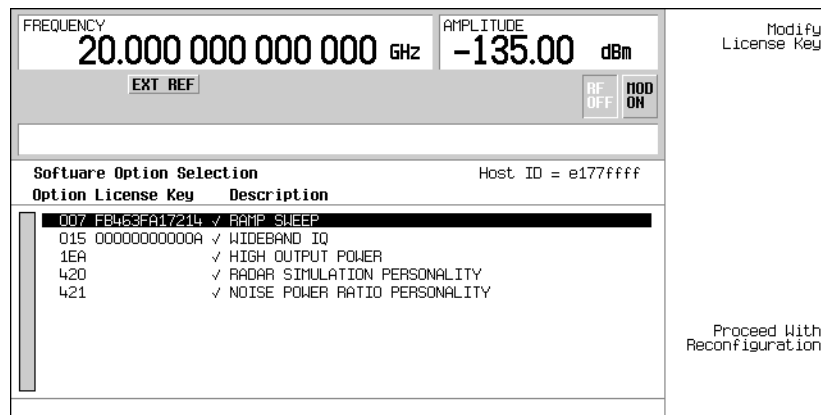
   Verify that the host ID shown on the display matches the host ID on the license key certificate. The host ID is a unique number for every instrument. If the host ID on the license key certificate does not match your instrument, the license key cannot enable the software option.

2. On the display is a list of software options that are already enabled (if any) and the software options that can be enabled. Some software options are linked to specific hardware options. Before a software option can be enabled, the appropriate hardware option must be installed. For example, Option 420, RADAR SIMULATION PERSONALITY, requires that Option 002/602, Internal Baseband Generator, be installed. If the software option that you intend to install is listed in a grey font, the required hardware may not be installed. (Look for an X in the "Selected" column of the appropriate hardware option in the Hardware Options menu.)

3. To enable the software option, highlight the desired option using the up/down arrow keys or the front panel knob.

4. Press **Modify License Key**. Enter the 12-character license key (from your license key certificate) using the softkeys and numeric keypad. When you have finished, press the **Enter** terminator softkey.

5. Press **Proceed With Reconfiguration** > **Confirm Change** to verify that you do want to reconfigure the signal generator with the options for which you have provided a license key. The instrument will enable the options and reboot.

3 Optimizing Performance

This chapter describes procedures that improve the performance of the Agilent PSG signal generator.

This chapter includes the following major sections:

Using External Leveling

The PSG signal generator can be externally leveled by connecting an external sensor at the point where leveled RF output power is desired. This sensor detects changes in RF output power and returns a compensating voltage to the signal generator's ALC input. The ALC circuitry raises or lowers (levels) the RF output power based on the voltage received from the external sensor, ensuring constant power at the point of detection.

There are two types of external leveling available on the PSG. You can use external leveling with a detector and coupler/power splitter setup, or a millimeter-wave source module.

To Level with Detectors and Couplers/Splitters

Figure 3-1 illustrates a typical external leveling setup. The power level feedback to the ALC circuitry is taken from the external negative detector, rather than the internal signal generator detector. This feedback voltage controls the ALC system, leveling the RF output power at the point of detection.

To use detectors and couplers/splitters for external leveling at an RF output frequency of 10 GHz and an amplitude of 0 dBm, follow the instructions in this section.
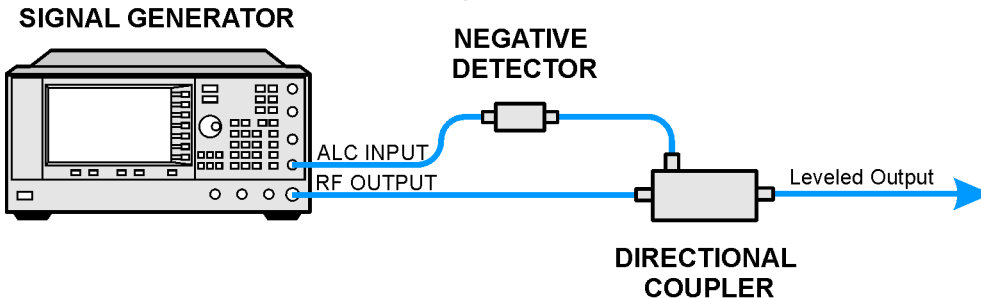
Required Equipment

- Agilent 8474E negative detector
- Agilent 87301D directional coupler
- cables and adapters, as required

Connect the Equipment

Set up the equipment as shown in Figure 3-1.

Figure 3-1   **External Detector Leveling with a Directional Coupler**

Configure the Signal Generator

1. Press **Preset**.

2. Press **Frequency** > **10** > **GHz**.

3. Press **Amplitude** > **0** > **dBm**.

4. Press **RF On/Off**.

5. Press **Leveling Mode** > **Ext Detector**.

   This deactivates the internal ALC detector and switches the ALC input path to the front panel ALC INPUT connector. The EXT indicator is activated in the AMPLITUDE area of the display.

NOTE   For signal generators with Option 1E1, notice that the ATTN HOLD (attenuator hold) annunciator is displayed. During external leveling, the signal generator automatically uncouples the attenuator from the ALC system for *all* external leveling points. While in this mode, RF output amplitude adjustment is limited to -20 to +25 dBm, the adjustment range of the ALC circuitry. For more information, see "External Leveling with Option 1E1 Signal Generators" on page 77.

---

6. Observe the coupling factor printed on the directional coupler at the detector port. Typically, this value is -10 to -20 dB.

   Enter the *positive* dB value of this coupling factor into the signal generator.

7. Press **More (1 of 2)** > **Ext Detector Coupling Factor** > **16** (or the positive representation of the value listed at the detector port of the directional coupler) > **dB**.

   Leveled output power is now available at the output of the directional coupler.


NOTE   While operating in external leveling mode, the signal generator's displayed RF output amplitude is affected by the coupling factor value, resulting in a calculated approximation of the actual RF output amplitude.

   To determine the actual RF output amplitude at the point of detection, measure the voltage at the external detector output and refer to Figure 3-2 or measure the power directly with a power meter.

Determining the Leveled Output Power

Figure 3-2 shows the input power versus output voltage characteristics for typical Agilent Technologies diode detectors. Using this chart, you can determine the leveled power at the diode detector input by measuring the external detector output voltage. You must then add the coupling factor to determine the leveled output power. The range of power adjustment is approximately -20 to +25 dBm.

Figure 3-2   **Typical Diode Detector Response at 25° C**
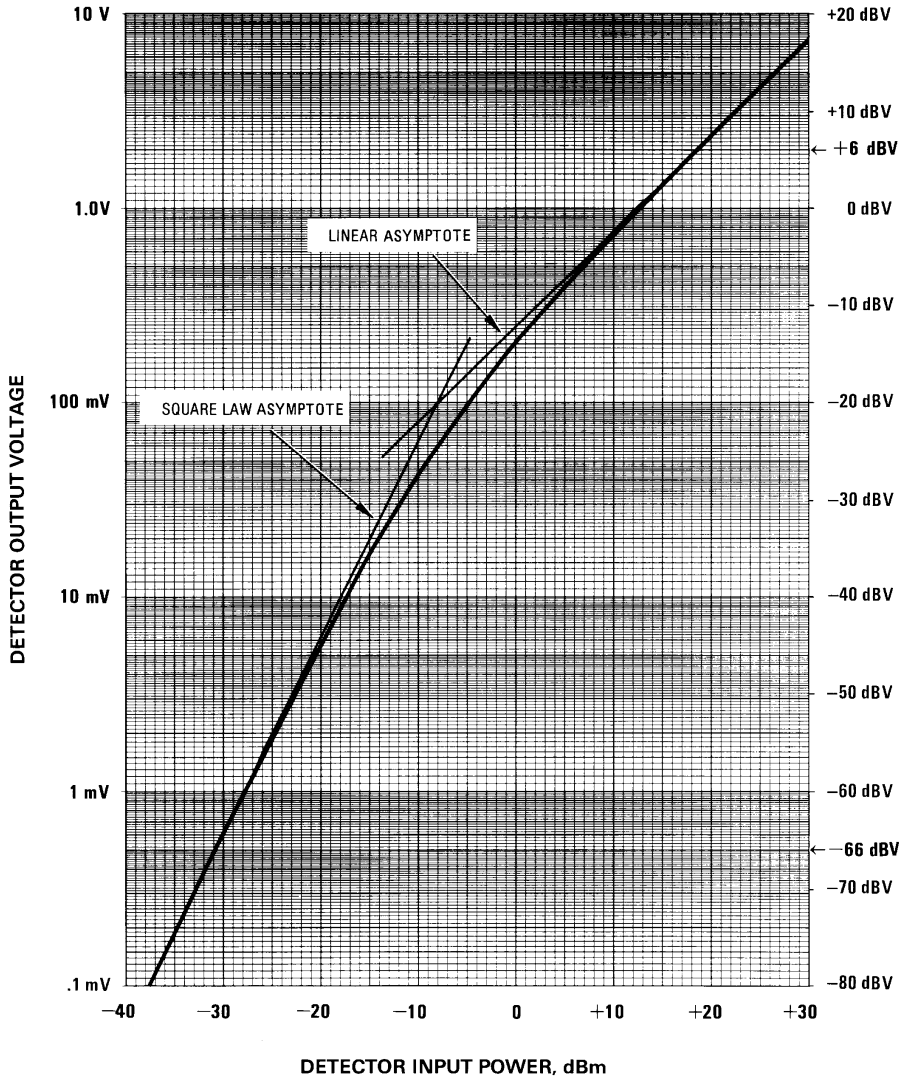
External Leveling with Option 1E1 Signal Generators

Signal generators with Option 1E1 contain a step attenuator prior to the RF output connector. During external leveling, the signal generator automatically holds the present attenuator setting (to avoid power transients that may occur during attenuator switching) as the RF amplitude is changed. A balance must be maintained between the amount of attenuation and the optimum ALC level to achieve the required RF output amplitude. For optimum accuracy and minimum noise, the ALC level should be greater than -10 dBm.

For example, leveling the CW output of a 30 dB gain amplifier to a level of -10 dBm requires the output of the signal generator to be approximately -40 dBm when leveled. This is beyond the amplitude limits of the ALC modulator alone, resulting in an unleveled RF output. Inserting 45 dB of attenuation results in an ALC level of +5 dBm, well within the range of the ALC modulator.

NOTE	In the example above, 55 dB is the preferred attenuation choice, resulting in an ALC level of +15 dBm. This provides adequate dynamic range for AM or other functions that vary the RF output amplitude.

To achieve the optimum ALC level at the signal generator RF output of -40 dBm for an unmodulated carrier, follow these steps:

1. Press **Amplitude > Set Atten > 45 > dB**.

2. Press **Set ALC Level > 5 > dBm**.

This sets the attenuator to 45 dB and the ALC level to +5 dBm, resulting in an RF output amplitude of -40 dBm, as shown in the AMPLITUDE area of the display.

To obtain flatness-corrected power, refer to "Creating and Applying User Flatness Correction" on page 78.

To Level with a mm-Wave Source Module

Millimeter-wave source module leveling is similar to external detector leveling. The power level feedback signal to the ALC circuitry is taken from the millimeter-wave source module, rather than the internal signal generator detector. This feedback signal levels the RF output power at the mm-wave source module output through the signal generator's rear panel SOURCE MODULE interface connector.

For instructions and setups, see "Extending the Frequency Range with a mm-Wave Source Module" on page 57.

Creating and Applying User Flatness Correction

User flatness correction allows the digital adjustment of RF output amplitude for up to 1601 frequency points in any frequency or sweep mode. Using an Agilent E4416A/17A or E4418B/19B power meter (controlled by the signal generator through GPIB) to calibrate the measurement system, a table of power level corrections is created for frequencies where power level variations or losses occur. These frequencies may be defined in sequential linear steps or arbitrarily spaced.

If you do not have an Agilent E4416A/17A or E4418B/19B power meter, or if your power meter does not have a GPIB interface, the correction values can be manually entered into the signal generator.

To allow different correction arrays for different test setups or different frequency ranges, you may save individual user flatness correction tables to the signal generator's memory catalog and recall them on demand.

Follow the steps in the next sections to create and apply user flatness correction to the signal generator's RF output.

Afterward, follow the steps in "Recalling and Applying a User Flatness Correction Array" on page 83 to recall a user flatness file from the memory catalog and apply it to the signal generator's RF output.

To Create a User Flatness Correction Array

In this example, you will create a user flatness correction array. The flatness correction array contains ten frequency correction pairs (amplitude correction values for specified frequencies), from 1 to 10 GHz in 1 GHz intervals.

An Agilent E4416A/17A/18B/19B power meter (controlled by the signal generator via GPIB) and E4413A power sensor are used to measure the RF output amplitude at the specified correction frequencies and transfer the results to the signal generator. The signal generator reads the power level data from the power meter, calculates the correction values, and stores the correction pairs in the user flatness correction array.

If you do not have the required Agilent power meter, or if your power meter does not have a GPIB interface, you can enter correction values manually.

Required Equipment

- Agilent E4416A/17A/18B/19B power meter

- Agilent E4413A E Series CW power sensor

- GPIB interface cable

- adapters and cables, as required

NOTE   If the setup has an external leveling configuration, the equipment setup in Figure 3-3
assumes that the steps necessary to correctly level the RF output have been followed. If you
have questions about external leveling, refer to "Using External Leveling" on page 74.

Configure the Power Meter

1. Select SCPI as the remote language for the power meter.

2. Zero and calibrate the power sensor to the power meter.

3. Enter the appropriate power sensor calibration factors into the power meter as appropriate.

4. Enable the power meter's cal factor array.

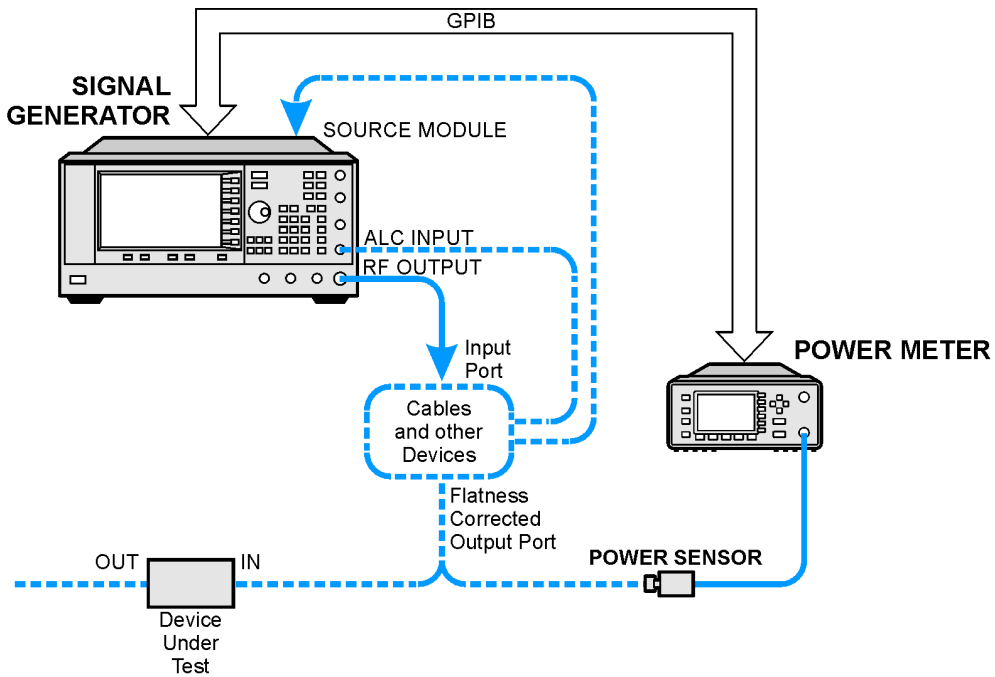NOTE   For operating information on your particular power meter/sensor, refer to its operating
guide.

Connect the Equipment

Connect the equipment as shown in Figure 3-3.

NOTE   During the process of creating the user flatness correction array, the power meter is slaved
              to the signal generator via GPIB. No other controllers are allowed on the GPIB interface.

Figure 3-3   **User Flatness Correction Equipment Setup**

Configure the Signal Generator

1. Press **Preset**.

2. Configure the signal generator to interface with the power meter.

   a. Press **Amplitude** > **More (1 of 2)** > **User Flatness** > **More (1 of 2)** > **Power Meter** > **E4416A**, **E4417A**, **E4418B**, or **E4419B**.

   b. Press **Meter Address** > enter the power meter's GPIB address > **Enter**.

   c. For E4417A and E4419B models, press **Meter Channel A B** to select the power meter's active channel.

   d. Press **Meter Timeout** to adjust the length of time before the instrument generates a timeout error if unsuccessfully attempting to communicate with the power meter.

3. Press **More (2 of 2)** > **Configure Cal Array** > **More (1 of 2)** > **Preset List** > **Confirm Preset**.

   This opens the User Flatness table editor and presets the cal array frequency/correction list.

4. Press **Configure Step Array**.

   This opens a menu for entering the user flatness step array data.

5. Press **Freq Start** > **1** > **GHz**.

6. Press **Freq Stop** > **10** > **GHz**.

7. Press **# of Points** > **10** > **Enter**.

   Steps 4, 5, and 6 enter the desired flatness-corrected frequencies into the step array.

8. Press **Return** > **Load Cal Array From Step Array** > **Confirm Load From Step Data**.

   This populates the user flatness correction array with the frequency settings defined in the step array.

9. Press **Amplitude** > **0** > **dBm**.

10. Press **RF On/Off**.

   This activates the RF output and the RF ON annunciator is displayed on the signal generator.

Perform the User Flatness Correction

NOTE   If you are not using an Agilent E4416A/17A/18B/19B power meter, or if your power meter does not have a GPIB interface, you can perform the user flatness correction manually. For instructions, see "Performing the User Flatness Correction Manually" on page 82.

---

1. Press **More (1 of 2)** > **User Flatness** > **Do Cal**.

    This creates the user flatness amplitude correction value table entries. The signal generator enters the user flatness correction routine and a progress bar is shown on the display.

2. Press **Done**.

    This loads the amplitude correction values into the user flatness correction array.

    If desired, press **Configure Cal Array**.

    This opens the user flatness correction array, where you can view the stored amplitude correction values. The user flatness correction array title displays User Flatness: (UNSTORED) indicating that the current user flatness correction array data has not been saved to the memory catalog.

Performing the User Flatness Correction Manually

If you are not using an Agilent E4416A/17A/18B/19B power meter, or if your power meter does not have a GPIB interface, complete the steps in this section and then continue with the user flatness correction tutorial.

1. Press **More (1 of 2)** > **User Flatness** > **Configure Cal Array**.

    This opens the User Flatness table editor and places the cursor over the frequency value
    (1 GHz) for row 1. The RF output changes to the frequency value of the table row containing the cursor and 1.000 000 000 00 is displayed in the AMPLITUDE area of the display.

2. Observe and record the measured value from the power meter.

3. Subtract the measured value from 0 dBm.

4. Move the table cursor over the correction value in row 1.

5. Press **Edit Item** > enter the difference value from step 3 > **dB**.

    The signal generator adjusts the RF output amplitude based on the correction value entered.

6. Repeat steps 2 through 5 until the power meter reads 0 dBm.

7. Use the down arrow key to place the cursor over the frequency value
    for the next row. The RF output changes to the frequency value of the table row containing the cursor, as shown in the AMPLITUDE area of the display.

8. Repeat steps 2 through 7 for every entry in the User Flatness table.

Save the User Flatness Correction Data to the Memory Catalog

This process allows you to save the user flatness correction data as in the signal generator's memory catalog. With several user flatness correction files saved to the memory catalog, any file can be recalled, loaded into the correction array, and applied to the RF output to satisfy specific RF output flatness requirements.

1. Press **Load/Store**.

2. Press **Store to File**.

3. Enter the file name FLATCAL1 using the alphanumeric softkeys, numeric keypad, or the knob.

4. Press **Enter**.

The user flatness correction array file FLATCAL1 is now stored in the memory catalog as a UFLT file.

Applying a User Flatness Correction Array

Press **Return** > **Return** > **Flatness Off On**.

This applies the user flatness correction array to the RF output. The UF indicator is activated in the AMPLITUDE section of the signal generator's display and the frequency correction data contained in the correction array is applied to the RF output amplitude.

Recalling and Applying a User Flatness Correction Array

Before performing the steps in this section, complete "To Create a User Flatness Correction Array" on page 78.

1. Press **Preset**.

2. Press **Amplitude** > **More (1 of 2)** > **User Flatness** > **Configure Cal Array** > **More (1 of 2)** > **Preset List** > **Confirm Preset**.

3. Press **More (2 of 2)** > **Load/Store**.

4. Ensure that the file FLATCAL1 is highlighted.

5. Press **Load From Selected File** > **Confirm Load From File**.

This populates the user flatness correction array with the data contained in the file FLATCAL1. The user flatness correction array title displays User Flatness: FLATCAL1.

6. Press **Return** > **Flatness Off On**.

This applies the user flatness correction data contained in FLATCAL1.

Returning the Signal Generator to GPIB Listener Mode

During the user flatness correction process, the power meter is slaved to the signal generator via GPIB, and no other controllers are allowed on the GPIB interface. The signal generator operates in GPIB talker mode, as a device controller for the power meter. In this operating mode, it cannot receive SCPI commands via GPIB.

If the signal generator is to be interfaced to a remote controller after performing the user flatness correction, its GPIB controller mode must be changed from GPIB talker to GPIB listener.

If an RF carrier has been previously configured, you must save the present instrument state before returning the signal generator to GPIB listener mode.

1. Save your instrument state to the instrument state register.

    For instructions, see "Saving an Instrument State" on page 68.

2. Press **GPIB Listener Mode**.

    This presets the signal generator and returns it to GPIB listener mode. The signal generator can now receive remote commands executed by a remote controller connected to the GPIB interface.

3. Recall your instrument state from the instrument state register.

    For instructions, see "Saving an Instrument State" on page 68.

To Create a User Flatness Correction Array with a mm-Wave Source Module

In this example, a user flatness correction array is created to provide flatness-corrected power at the output of an Agilent 83554A millimeter-wave source module driven by an E8247C signal generator.

The flatness correction array contains 28 frequency correction pairs (amplitude correction values for specified frequencies), from 26.5 to 40 GHz in 500 MHz intervals. This will result in 28 evenly spaced flatness corrected frequencies between 26.5 GHz and 40 GHz at the output of the 83554A millimeter-wave source module.

An Agilent E4416A/17A/18B/19B power meter (controlled by the signal generator via GPIB) and R8486A power sensor are used to measure the RF output amplitude of the millimeter-wave source module at the specified correction frequencies and transfer the results to the signal generator. The signal generator reads the power level data from the power meter, calculates the correction values, and stores the correction pairs in the user flatness correction array.

If you do not have the required Agilent power meter, or if your power meter does not have a GPIB interface, you can enter correction values manually.

Required Equipment

- Agilent 83554A millimeter-wave source module

- Agilent E4416A/17A/18B/19B power meter

- Agilent R8486A power sensor

- Agilent 8349B microwave amplifier (required for signal generators without Option 1EA)

- GPIB interface cable

- adapters and cables as required

NOTE   The equipment setups in Figure 3-4 and Figure 3-5 assume that the steps necessary to correctly level the RF output have been followed. If you have questions about leveling with a millimeter-wave source module, refer to "To Level with a mm-Wave Source Module" on page 77.

Configure the Power Meter

1. Select SCPI as the remote language for the power meter.

2. Zero and calibrate the power sensor to the power meter.

3. Enter the appropriate power sensor calibration factors into the power meter as appropriate.

4. Enable the power meter's cal factor array.

NOTE   For operating information on your particular power meter/sensor, refer to their operating guides.

Connect the Equipment

CAUTION   To prevent damage to the signal generator, turn off the line power to the signal generator before connecting the source module interface cable to the rear panel SOURCE MODULE interface connector.

---

1. Turn off the line power to the signal generator.

2. Connect the equipment. For standard signal generators, use the setup in Figure 3-4. For Option 1EA signal generators, use the setup in Figure 3-5.

NOTE	During the process of creating the user flatness correction array, the power meter is slaved to the signal generator via GPIB. No other controllers are allowed on the GPIB interface.

Figure 3-4   **User Flatness with mm-Wave Source Module for a Signal Generator without Option 1EA**
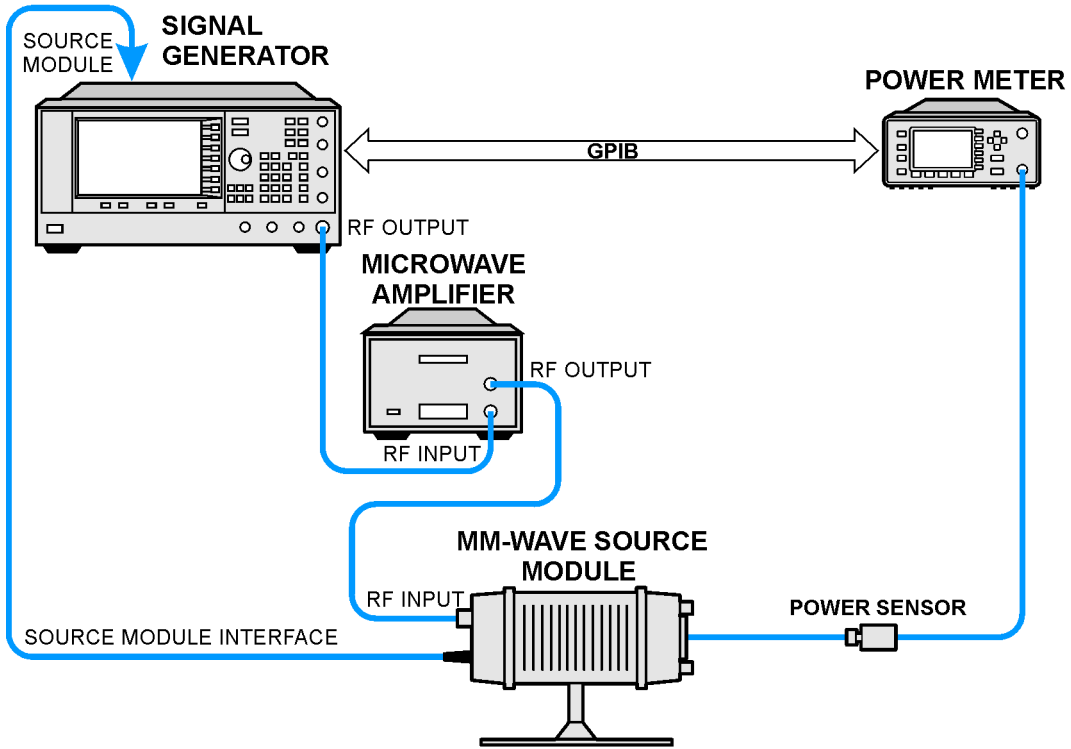
Figure 3-5   **User Flatness with mm-Wave Source Module and Option 1EA Signal Generator**
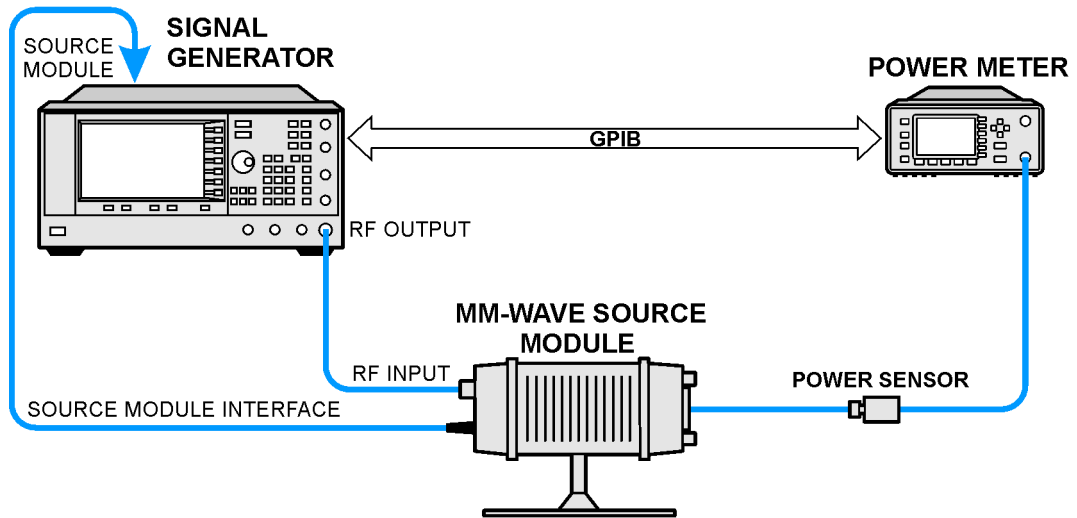
pe920a

NOTE   To ensure adequate RF amplitude at the mm-wave source module RF input when using
                 Option 1EA signal generators, maximum amplitude loss through the adapters and cables
                 connected between the signal generator's RF output and the mm-wave source module's RF
                 input should be less than 1.5 dB.

Configure the Signal Generator

1. Turn on the signal generator's line power.

   Upon power-up, the signal generator automatically does the following:

   • senses the mm-wave source module

   • switches the signal generator's leveling mode to external/source module

   • sets the mm-wave source module frequency and amplitude to the source module's preset values

   • displays the RF output frequency and amplitude values available at the mm-wave source module
     output

   The `MMMOD` indicator in the `FREQUENCY` area and the `MM` indicator in the `AMPLITUDE` area of the signal
   generator's display indicate that the mm-wave source module is active

---

NOTE	Refer to the mm-wave source module specifications for the specific frequency and amplitude ranges.

---

2. Configure the signal generator to interface with the power meter.

   a. Press **Amplitude** > **More (1 of 2)** > **User Flatness** > **More (1 of 2)** > **Power Meter** > **E4416A**, **E4417A**, **E4418B**, or **E4419B**.

   b. Press **Meter Address** > enter the power meter's GPIB address > **Enter**.

   c. For E4417A and E4419B models, press **Meter Channel A B** to select the power meter's active channel.

   d. Press **Meter Timeout** to adjust the length of time before the instrument generates a timeout error if unsuccessfully attempting to communicate with the power meter.

3. Press **More (2 of 2)** > **Configure Cal Array** > **More (1 of 2)** > **Preset List** > **Confirm Preset**.

   This opens the User Flatness table editor and resets the cal array frequency/correction list.

4. Press **Configure Step Array**.

   This opens a menu for entering the user flatness step array data.

5. Press **Freq Start** > **26.5** > **GHz**.

6. Press **Freq Stop** > **40** > **GHz**.

7. Press **# of Points** > **28** > **Enter**.

   This enters the desired flatness-corrected frequencies (26.5 GHz to 40 GHz in 500 MHz intervals) into the step array.

8. Press **Return** > **Load Cal Array From Step Array** > **Confirm Load From Step Data**.

   This populates the user flatness correction array with the frequency settings defined in the step array.

9. Press **Amplitude** > **0** > **dBm**.

10. Press **RF On/Off**.

   This activates the RF output and the RF ON annunciator is displayed on the signal generator.

Perform the User Flatness Correction

NOTE	If you are not using an Agilent E4416A/17A/18B/19B power meter, or if your power meter does not have a GPIB interface, you can perform the user flatness correction manually. For instructions, see Performing the User Flatness Correction Manually below.

---

1. Press **More (1 of 2)** > **User Flatness** > **Do Cal**.

   This creates the user flatness amplitude correction value table entries. The signal generator begins the user flatness correction routine and a progress bar is shown on the display.

2. When prompted, press **Done**.

   This loads the amplitude correction values into the user flatness correction array.

   If desired, press **Configure Cal Array**.

   This opens the user flatness correction array, where you can view the list of defined frequencies and their calculated amplitude correction values. The user flatness correction array title displays User Flatness: (UNSTORED) indicating that the current user flatness correction array data has not been saved to the memory catalog.

Performing the User Flatness Correction Manually

If you are not using an Agilent E4416A/17A/18B/19B power meter, or if your power meter does not have a GPIB interface, complete the steps in this section and then continue with the user flatness correction tutorial.

1. Press **More (1 of 2)** > **User Flatness** > **Configure Cal Array**.

   This opens the User Flatness table editor and places the cursor over the frequency value (26.5 GHz) for row 1. The RF output changes to the frequency value of the table row containing the cursor and 26.500 000 000 00 is displayed in the AMPLITUDE area of the display.

2. Observe and record the measured value from the power meter.

3. Subtract the measured value from 0 dBm.

4. Move the table cursor over the correction value in row 1.

5. Press **Edit Item** > enter the difference value from step 3 > **dB**.

   The signal generator adjusts the RF output amplitude based on the correction value entered.

6. Repeat steps 2 through 5 until the power meter reads 0 dBm.

7. Use the down arrow key to place the cursor over the frequency value for the next row. The RF output changes to the frequency value highlighted by the cursor, as shown in the AMPLITUDE area of the display.

8. Repeat steps 2 through 7 for every entry in the User Flatness table.

Save the User Flatness Correction Data to the Memory Catalog

This process allows you to save the user flatness correction data as a file in the signal generator's memory catalog. With several user flatness correction files saved to the memory catalog, specific files can be recalled, loaded into the correction array, and applied to the RF output to satisfy various RF output flatness requirements.

1. Press **Load/Store**.

2. Press **Store to File**.

3. Enter the file name FLATCAL2 using the alphanumeric softkeys and the numeric keypad.

4. Press **Enter**.

The user flatness correction array file FLATCAL2 is now stored in the memory catalog as a UFLT file.

Applying the User Flatness Correction Array

1. Press **Return** > **Return** > **Flatness Off On**.

   This applies the user flatness correction array to the RF output. The UF indicator is activated in the AMPLITUDE section of the signal generator's display and the frequency correction data contained in the correction array is applied to the RF output amplitude of the mm-wave source module.

Recalling and Applying a User Flatness Correction Array

Before performing the steps in this section, complete the section "To Create a User Flatness Correction Array with a mm-Wave Source Module" on page 84.

1. Press **Preset**.

2. Press **Amplitude** > **More (1 of 2)** > **User Flatness** > **Configure Cal Array** > **More (1 of 2)** > **Preset List** > **Confirm Preset**.

3. Press **More (2 of 2)** > **Load/Store**.

4. Ensure that the file FLATCAL2 is highlighted.

5. Press **Load From Selected File** > **Confirm Load From File**.

   This populates the user flatness correction array with the data contained in the file FLATCAL2. The user flatness correction array title displays User Flatness: FLATCAL2.

6. Press **Return** > **Flatness Off On**.

   This activates flatness correction using the data contained in the file FLATCAL2.

Selecting ALC Bandwidth

For internal leveling, the signal generator uses automatic leveling control (ALC) circuitry prior to the RF output. ALC bandwidth has five selections: automatic, 100 Hz, 1 kHz, 10 kHz, and 100 kHz.

At signal generator preset, the ALC bandwidth selection is set to Auto. In this configuration, the signal generator automatically adjusts the ALC bandwidth between three of the four possible settings, depending on which functions are currently active. Figure 3-6 shows the signal generator's automatic ALC bandwidth selection decision tree.

Figure 3-6   **Decision Tree for Automatic ALC Bandwidth Selection**
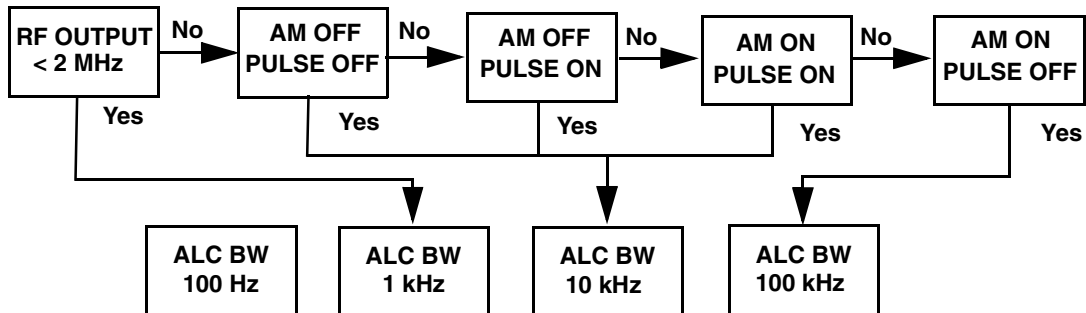
To Select an ALC Bandwidth

Press **Amplitude** > **ALC BW** > **100 Hz**, **1 kHz**, **10 kHz**, or **100 kHz**.

This overrides the automatic ALC bandwidth selection with your specific selection.

4   Analog Modulation

This chapter describes the analog modulation capability in Agilent E8257C PSG Analog and E8267C PSG Vector Signal Generators.

This chapter includes the following major sections:

Analog Modulation Waveforms

The signal generator can modulate the RF carrier with four types of analog modulation: amplitude, frequency, phase, and pulse.

Available internal waveforms include:

Sine   sine wave with adjustable amplitude and frequency

Dual-Sine   dual-sine waves with individually adjustable frequencies and a percent-of-peak-amplitude setting for the second tone (available from function generator only)

Swept-Sine   swept-sine wave with adjustable start and stop frequencies, sweep rate, and sweep trigger settings (available from function generator only)

Triangle   triangle wave with adjustable amplitude and frequency

Ramp   ramp with adjustable amplitude and frequency

Square   square wave with adjustable amplitude and frequency

Noise   noise with adjustable amplitude generated as a peak-to-peak value (RMS value is approximately 80% of the displayed value)

Configuring AM

In this example, you will learn how to generate an amplitude-modulated RF carrier.

To Set the Carrier Frequency

1. Press **Preset**.

2. Press **Frequency** > **1340** > **kHz**.

To Set the RF Output Amplitude

Press **Amplitude** > **0** > **dBm**.

To Set the AM Depth and Rate

1. Press the **AM** hardkey.

2. Press **AM Depth** > **90** > **%**.

3. Press **AM Rate** > **10** > **kHz**.

The signal generator is now configured to output a 0 dBm, amplitude-modulated carrier at 1340 kHz with the AM depth set to 90% and the AM rate set to 10 kHz. The shape of the waveform is a sine wave. Notice that sine is the default selection for the **AM Waveform** softkey, which can be viewed by pressing **(More 1 of 2)**.

To Turn on Amplitude Modulation

Follow these remaining steps to output the amplitude-modulated signal.

1. Press the **AM Off On** softkey to On.

2. Press the front panel **RF On Off** key.

The AM and RF ON annunciators are now displayed. This indicates that you have enabled amplitude modulation and the signal is now being transmitted from the RF OUTPUT connector.

Configuring FM

In this example, you will learn how to create a frequency-modulated RF carrier.

To Set the RF Output Frequency

1. Press **Preset**.

2. Press **Frequency** > **1** > **GHz**.

To Set the RF Output Amplitude

Press **Amplitude** > **0** > **dBm.**

To Set the FM Deviation and Rate

1. Press the **FM/ΦM** hardkey.

2. Press **FM Dev** > **75** > **kHz**.

3. Press **FM Rate** > **10** > **kHz**.

The signal generator is now configured to output a 0 dBm, frequency-modulated carrier at 1 GHz with a 75 kHz deviation and a 10 kHz rate. The shape of the waveform is a sine wave. (Notice that sine is the default for the **FM Waveform** softkey. Press **More (1 of 2)** to see the softkey.)

To Activate FM

1. Press **FM Off On** to On.

2. Press **RF On/Off**.

The FM and RF ON annunciators are now displayed. This indicates that you have enabled frequency modulation and the signal is now being transmitted from the RF OUTPUT connector.

Configuring ΦM

In this example, you will learn how to create a phase-modulated RF carrier.

To Set the RF Output Frequency

1. Press **Preset**.

2. Press **Frequency** > **3** > **GHz**.

To Set the RF Output Amplitude

Press **Amplitude** > **0** > **dBm**.

To Set the FM Deviation and Rate

1. Press the **FM/ΦM** hardkey.

2. Press the **FM ΦM** softkey.

3. Press **FM Dev** > **.25** > **pi rad**.

4. Press **FM Rate** > **10** > **kHz**.

The signal generator is now configured to output a 0 dBm, phase-modulated carrier at 3 GHz with a 0.25 p radian deviation and 10 kHz rate. The shape of the waveform is a sine wave. (Notice that sine is the default for the **FM Waveform** softkey. Press **More (1 of 2)** to see the softkey.)

To Activate FM

1. Press **FM Off On**.

2. Press **RF On/Off**.

The FM and RF ON annunciators are now displayed. This indicates that you have enabled phase modulation and the signal is now being transmitted from the RF OUTPUT connector.

Configuring Pulse Modulation

In this example, you will learn how to create a pulse-modulated RF carrier.

To Set the RF Output Frequency

1. Press **Preset**.

2. Press **Frequency** > **2** > **GHz**.

To Set the RF Output Amplitude

Press **Amplitude** > **0** > **dBm**.

To Set the Pulse Period and Width

1. Press **Pulse** > **Pulse Period** > **100** > **usec**.

2. Press **Pulse** > **Pulse Width** > **24** > **usec**.

The signal generator is now configured to output a 0 dBm, pulse-modulated carrier at 2 GHz with a 100-microsecond pulse period and 24-microsecond pulse width. The pulse source is set to Internal Free Run. (Notice that Internal Free Run is the default for the **Pulse Source** softkey.)

To Activate Pulse Modulation

Follow these remaining steps to output the pulse-modulated signal.

1. Press **Pulse Off On** to On.

2. Press **RF On/Off**.

The Pulse and RF ON annunciators are now displayed. This indicates that you have enabled pulse modulation and the signal is now being transmitted from the RF OUTPUT connector.

Configuring the LF Output

The signal generator has a low frequency (LF) output. The LF output's source can be switched between **Internal 1 Monitor**, **Internal 2 Monitor**, **Function Generator 1**, or **Function Generator 2**.

Using **Internal 1 Monitor** or **Internal 2 Monitor** as the LF output source, the LF output provides a replica of the signal from the internal source that is being used to modulate the RF output. The specific modulation parameters for this signal are configured through the AM, FM, or FM menus.

Using **Function Generator 1** or **Function Generator 2** as the LF output source, the function generator section of the internal modulation source drives the LF output directly. Frequency and waveform are configured from the LF output menu, not through the AM, FM, or FM menus. You can select the waveform shape from the following choices:

Sine	sine wave with adjustable amplitude and frequency
Dual-Sine	dual-sine waves with individually adjustable frequencies and a percent-of-peak-amplitude setting for the second tone (available from function generator 1 only)
Swept-Sine	a swept-sine wave with adjustable start and stop frequencies, sweep rate, and sweep trigger settings (available from function generator 1 only)
Triangle	triangle wave with adjustable amplitude and frequency
Ramp	ramp with adjustable amplitude and frequency
Square	square wave with adjustable amplitude and frequency
Noise	noise with adjustable amplitude generated as a peak-to-peak value (RMS value is approximately 80% of the displayed value)
DC	direct current with adjustable amplitude

NOTE   The **LF Out Off On** softkey controls the operating state of the LF output. However when the LF output source selection is **Internal Monitor**, you have three ways of controlling the output. You can use the modulation source (AM, FM, or FM) on/off key, the LF output on/off key, or the **Mod On/Off** softkey.

The **RF On/Off** hardkey does not apply to the LF OUTPUT connector.

To Configure the LF Output with an Internal Modulation Source

In this example, the internal FM modulation is the LF output source.

NOTE   Internal modulation (**Internal Monitor**) is the default LF output source.

Configuring the Internal Modulation as the LF Output Source

1. Press **Preset**.

2. Press the **FM/ΦM** hardkey.

3. Press **FM Dev > 75 > kHz**.

4. Press **FM Rate > 10 > kHz**.

5. Press **FM Off On**.

You have set up the FM signal with a rate of 10 kHz and 75 kHz of deviation. The FM annunciator is activated indicating that you have enabled frequency modulation.

Configuring the Low Frequency Output

1. Press the **LF Out** hardkey.

2. Press **LF Out Amplitude > 3 > Vp**.

3. Press **LF Out Off On**.

You have configured the LF output signal for a 3 volt sine wave (default wave form) output which is frequency modulated using the **Internal 1 Monitor** source selection (default source).

To Configure the LF Output with a Function Generator Source

In this example, the function generator is the LF output source.

Configuring the Function Generator as the LF Output Source

1. Press **Preset**.

2. Press the **LF Out** hardkey.

3. Press **LF Out Source > Function Generator 1**.

Configuring the Waveform

1. Press **LF Out Waveform** > **Swept-Sine**.

2. Press **LF Out Start Freq** > **100** > **Hz**.

3. Press **LF Out Stop Freq** > **1** > **kHz**.

4. Press **Return** > **Return**.

   This returns you to the top LF Output menu.

Configuring the Low Frequency Output

1. Press **LF Out Amplitude** > **3** > **Vp**.

   This sets the LF output amplitude to 3 Vp.

2. Press **LF Out Off On**.

The LF output is now transmitting a signal using Function Generator 1 that is providing a
3 Vp swept-sine waveform. The waveform is sweeping from 100 Hz to 1 kHz.

5 Custom Arb Waveform Generator

This chapter describes the Custom Arb Waveform Generator mode which is available only in E8267C PSG vector signal generators.

This chapter includes the following major sections:

- "Overview of Using Custom Arb Waveform Generator Mode" on page 104
- "Working with Predefined Modes" on page 105
- "Working with Filters" on page 110
- "Working with Symbol Rates" on page 120
- "Working with Modulation Types" on page 124
- "Working with Configuration of Hardware" on page 126

Overview of Using Custom Arb Waveform Generator Mode

Custom Arb Waveform Generator mode can produce a single modulated carrier or multiple modulated carriers. Each modulated carrier waveform must be calculated and generated before it can be output; this signal generation occurs on the internal baseband generator (Option 002/602). Once a waveform has been created, it can be stored and recalled which enables repeatable playback of test signals.

To begin using the Custom Arb Waveform Generator mode, select whether to create a single modulated carrier or a multiple modulated carrier waveform:

- If you want to create a single modulated carrier waveform, start by selecting a Digital Modulation Setup from a set of predefined modes (setups). Once a predefined mode is selected, you can modify the Modulation Type, the Filter being used, the Symbol Rate, and the type of Triggering; the Data Pattern is random by default. This modified setup can then be stored and reused.

- If you want to create a multiple modulated carrier waveform, start by selecting a Multicarrier Setup from a set of predefined modes (setups). Once a predefined mode is selected, you can modify the number of carriers to be created, the frequency spacing between each carrier, whether the phase offset between each carrier is to be fixed or random, and the type of Triggering; the Data Pattern is random by default, the Filter is set to 40 MHz by default, and the Symbol Rate is automatically specified by the selected Modulation Type being used.

Working with Predefined Modes

In this section, you will learn about the following:

- Using Predefined Mode

  When you select a predefined mode, default values for components of the setup (such as the data pattern, filter, symbol rate, modulation type, and the burst shape) are automatically specified.

- Using User-Defined Mode

To Select a Predefined Mode or Custom Digital Mod State

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Arb Waveform Generator**.

3. Press **Setup Select** > select one of the following:

   - Either press one of the predefined modes: **NADC**, **PDC**, **PHS**, **GSM**, **DECT**, **EDGE**, **APCO 25 w/C4FM**, **APCO 25 w/CQPSK**, **CDPD**, **PWT**, or **TETRA**.

     This selects a predefined mode where filtering, symbol rate, and modulation type are defined by the predefined mode that you selected and returns you to the top-level custom modulation menu; it does not include bursting or channel coding.

   - or press **Custom Digital Mod State**

     This selects a custom setup from the Catalog of DMOD Files displayed; these are files that you would have previously created by modifying a predefined mode and then saved to the Memory Catalog.

To Select a Predefined Mode (EDGE Example)

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator**.

3. Press **Setup Select** > **EDGE**.

4. Press **Digital Modulation Off On**.

   This generates a waveform with the predefined EDGE state; the display changes to Dig Mod Setup: EDGE. During waveform generation, the DIGMOD and I/Q annunciators appear and the predefined digital modulation state is stored in volatile memory.

5. Set the RF output frequency to 891 MHz.

6. Set the output amplitude to −5 dBm.

7. Press **RF On/Off**.

   The predefined EDGE waveform should be available at the signal generator's RF OUTPUT connector.

To Select a User-Defined Single-Carrier Setup

In this procedure, you learn how to start with a single-carrier NADC digital modulation and modify it to a custom waveform with customized modulation type, symbol rate, and filtering.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Setup Select** > **NADC**.

3. Press **Digital Mod Define** > **Modulation Type** > **PSK** > **QPSK and OQPSK** > **QPSK**.

4. Press **Symbol Rate** > **56** > **ksps**.

5. Press **Filter** > **Select** > **Nyquist**.

6. Press **Return** > **Return**.

7. Press **Digital Modulation Off On**.

   This generates a waveform with the custom single-carrier NADC digital modulation state. The display changes to Dig Mod Setup: NADC (Modified). During waveform generation, the DIGMOD and I/Q annunciators appear and the custom single-carrier digital modulation state is stored in volatile memory.

8. Set the RF output frequency to 835 MHz.

9. Set the output amplitude to 0 dBm.

10. Press **RF On/Off**.

    The user-defined NADC signal is now available at the RF OUTPUT connector.

11. Press **Return** > **Return**.

    This returns to the top-level Digital Modulation menu, where **Digital Modulation Off On** is the first softkey.

12. Press **Digital Mod Define** > **Store Custom Dig Mod State** > **Store To File**.

    If there is already a file name from the Catalog of DMOD Files occupying the active entry area, press the following keys:

    Edit Keys > **Clear Text**

13. Enter a file name (for example, NADCQPSK) using the alpha keys and the numeric keypad.

14. Press **Enter**.

    The user-defined single-carrier digital modulation state should now be stored in non-volatile memory. The RF output amplitude, frequency, and operating state settings are not stored as part of a user-defined digital modulation state file.

To Select a User-Defined Multicarrier EDGE Setup

In this procedure, you learn how to customize a predefined multicarrier digital modulation setup by creating a custom 3-carrier EDGE digital modulation state.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Arb Waveform Generator**.

3. Press **Multicarrier Off On**.

4. Press **Multicarrier Define** > **Initialize Table** > **Carrier Setup** > **EDGE** > **Done**.

5. Highlight the Freq Offset value (500.000 kHz) for the carrier in row 2.

6. Press **Edit Item** > **–625** > **kHz**.

7. Highlight the Power value (0.00 dB) for the carrier in row 2.

8. Press **Edit Item** > **–10** > **dB**.

    You should have a custom 2-carrier EDGE waveform with a carrier at a frequency offset of –625 kHz and a power level of –10.00 dBm, as shown in the following figure.
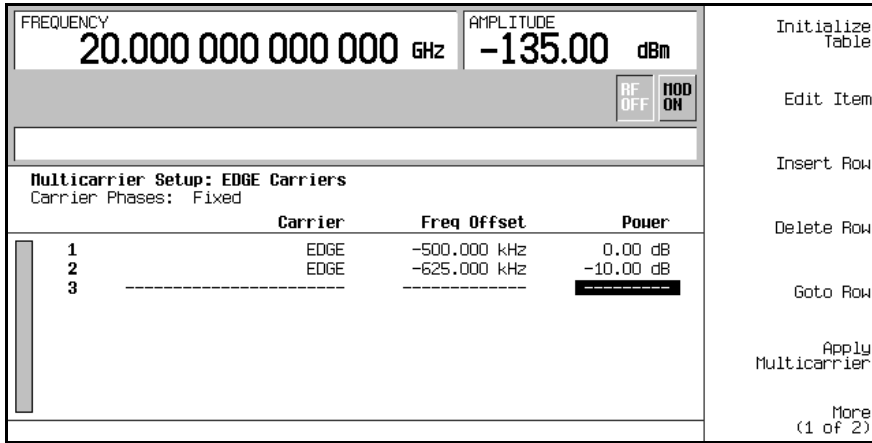
9. Press **Return** > **Digital Modulation Off On**.

    This generates a waveform with the custom multicarrier EDGE state. The display changes to `Dig Mod Setup: Multicarrier (Modified)`. During waveform generation, the `DIGMOD` and `I/Q` annunciators appear and the new custom multicarrier EDGE state is stored in volatile memory.

10. Set the RF output frequency to 890.01 MHz.

11. Set the output amplitude to –10 dBm.

12. Press **RF On/Off**.

    The custom multicarrier EDGE waveform should be available at the RF OUTPUT connector; it does not include bursting or channel coding.

13. Press **Mode** > **Custom** > **Arb Waveform Generator**, where **Digital Modulation Off On** is the first softkey.

14. Press **Multicarrier Off On** > **Multicarrier Define** > **More (1 of 2)** > **Load/ Store** > **Store To File**.

    If there is already a file name from the `Catalog of MDMOD Files` occupying the active entry area, press **Edit Keys** > **Clear Text**.

15. Enter a file name (for example, `EDGEM1`) using the alpha keys and the numeric keypad.

16. Press **Enter**.

The user-defined multicarrier digital modulation state is now stored in non-volatile memory.

NOTE	The RF output amplitude, frequency, and operating state settings (such as RF On/Off) are not stored as part of a user-defined digital modulation state file. For more information, refer to "Using Data Storage Functions" on page 66.

To Recall a User-Defined Custom Digital Mod State

In this procedure, you learn how to select (recall) a Custom Digital Mod State from the Memory Catalog. The custom modulation state had to be previously stored in the `Catalog of DMOD Files`.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Setup Select**.

3. Press **More (1 of 2)** > **Custom Digital Mod State**.

4. Press **Select File** to select a custom modulation state from the `Catalog of DMOD Files`.

The user-defined custom digital modulation state should now be recalled from non-volatile memory. Because the RF output amplitude, frequency, and operating state settings are not stored as part of a user-defined digital modulation state file, they must still be set or recalled separately. For more information, refer to "Using Data Storage Functions" on page 66.

Working with Filters

In this section, you will learn about the following:

- "Understanding FIR Filters" on page 110

- Using a Predefined FIR Filter

  - "To Select a Predefined Root Nyquist, Nyquist, or Gaussian Filter" on page 111
  - "To Adjust the Filter Alpha of a Predefined Root Nyquist or Nyquist Filter" on page 112
  - "To Adjust the Bandwidth-Bit-Time (BbT) Product of a Predefined Gaussian Filter" on page 112
  - "To Select a Predefined Rectangle Filter" on page 112
  - "To Select an APCO 25-Specified C4FM Filter" on page 112
  - "To Restore Default FIR Filter Parameters" on page 112

- Using a User-Defined FIR Filter

  FIR filters can be created and modified by defining the FIR coefficients or by defining the oversample ratio (number of filter coefficients per symbol) to be applied to your own custom FIR filter.

  - "To Modify Predefined FIR Coefficients for a Gaussian Filter with the FIR Values Editor" on page 113
  - "To Create a User-Defined FIR Filter with the FIR Values Editor" on page 115

Understanding FIR Filters

Use the signal generator's **Filter** menu selections to apply a filter to the generated signal, define a finite impulse response (FIR) filter, change a Root Nyquist or Nyquist filter alpha, change a Gaussian filter BbT, or restore all filter parameters to their default state.

NOTE   The procedures in this section work only for filters created in Custom Arb Waveform Generator mode; they do not work with downloaded files, such as those created in Matlab.

Predefined Filters (Filter > Select)

- **Root Nyquist** is a root-raised cosine pre-modulation FIR filter.

  Use a Root Nyquist filter when you want to place half of the filtering in the transmitter and the other half in the receiver. The ideal root-raised cosine filter frequency response has unity gain at low frequencies, the square root of raised cosine function in the middle, and total attenuation at high frequencies. The width of the middle frequencies is defined by the roll off factor or Filter Alpha (0 < Filter Alpha < 1).

- **Nyquist** is a raised cosine pre-modulation FIR filter.

  You can use a Nyquist filter to reduce the bandwidth required by a signal without losing information. The ideal raised cosine filter frequency response comprises unity gain at low frequencies, a raised cosine function in the middle, and total attenuation at high frequencies. The width of the middle frequencies is defined by the roll off factor or Filter Alpha (0 < Filter Alpha < 1).

- **Gaussian** is a Gaussian pre-modulation FIR filter.

- **User FIR** enables you to select from a Catalog of FIR filters; use this selection if the other predefined FIR filters do not meet your filtering needs. For more information, see **Define User FIR**, below.

- **Rectangle** is a rectangular pre-modulation FIR filter.

- **APCO 25 C4FM** is an APCO 25-specified C4FM filter; this is a Nyquist filter with an alpha of 0.200 that is combined with a shaping filter.

Filter Parameters

- **Filter Alpha** enables you to adjust the filter alpha for a Nyquist or root Nyquist filter. If a Gaussian filter is used, you will see Filter BbT; this softkey is grayed out when any other filter is selected.

- **Define User FIR** is available for when the predefined FIR filters do not meet your filtering needs. You can define FIR coefficients and set the oversample ratio (number of filter coefficients per symbol) to be applied to a custom FIR filter.

- **Restore Default Filters** replaces the current FIR filter with the default FIR filter for the selected format.

To Select a Predefined Root Nyquist, Nyquist, or Gaussian Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Select** and select one of the following: **Root Nyquist**, **Nyquist**, or **Gaussian**.

To Adjust the Filter Alpha of a Predefined Root Nyquist or Nyquist Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Filter Alpha**.

3. Enter a new Filter Alpha value and press **Enter**.

To Adjust the Bandwidth-Bit-Time (BbT) Product of a Predefined Gaussian Filter

1. Press **Filter** > **Select** > **Gaussian**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Select** > **Gaussian**.

3. Press **Filter BbT**.

4. Enter a new Bandwidth-Bit-Time (BbT) product filter parameter and press **Enter**.

To Select a Predefined Rectangle Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Select** > **More (1 of 2)** > **Rectangle**.

To Select an APCO 25-Specified C4FM Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Select** > **(More 1 of 2)** > **APCO 25 C4FM**.

   This selects a Nyquist filter with an alpha of 0.200 which is combined with a shaping filter.

To Restore Default FIR Filter Parameters

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Filter** > **Restore Default Filter**.

   This replaces the current FIR filter with the default filter for the selected modulation format.

To Modify Predefined FIR Coefficients for a Gaussian Filter with the FIR Values Editor

You can define from 1 to 32 FIR coefficients, where the maximum combination of symbols and oversample ratio is 1024 coefficients.

The FIR Values editor allows a maximum filter length of 1024 coefficients, but the PSG hardware is limited to 512 symbols for arbitrary waveform generation. The number of symbols equals the number of coefficients divided by the oversample ratio. If you enter more than 512 symbols for arbitrary waveform generation, the PSG cannot use the filter; it will decimate the filter (throw away coefficients) until this condition is met. It will use the filter, but fine resolution may be lost from the impulse response.

FIR filters stored in signal generator memory can easily be modified using the FIR Values editor. In this example, you will load the FIR Values editor with coefficient values from a default FIR filter (or, if one has been defined, a user-defined FIR file that has been stored in the Memory Catalog), modify the coefficient values, and store the new file to the Memory Catalog.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Arb Waveform Generator** > **Digital Mod Define** > **Filter**.

3. Press **Define User FIR** > **More (1 of 2)** > **Load Default FIR** > **Gaussian**.

4. Press **Filter BbT** > **0.300** > **Enter**.

5. Press **Filter Symbols** > **8** > **Enter**.

6. Press **Generate**.

NOTE   The actual oversample ratio during modulation is automatically selected by the instrument. A value between 4 and 16 is chosen dependent on the symbol rate, the number of bits per symbol of the modulation type, and the number of symbols.

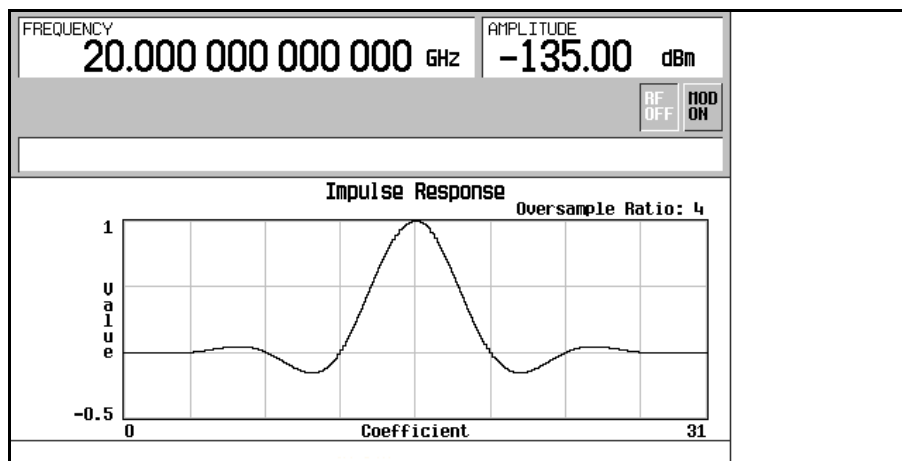
7. Press **Display Impulse Response**.

   You should see a graph that shows the impulse response of the current set of FIR coefficients.

Figure 5-1

8. Press **Return**.

9. Highlight coefficient 15.

10. Press **0** > **Enter**.

11. Press **Display Impulse Response**.

Figure 5-2

The graphic display can provide a useful troubleshooting tool (in this case, it indicates that a coefficient value is set incorrectly, resulting in an improper Gaussian response).

12. Press **Return**.

13. Highlight coefficient 15.

14. Press **1** > **Enter**.

15. Press **Load/Store** > **Store To File**.

16. Name the file NEWFIR2.

17. Press **Enter**.

The contents of the current FIR Values editor are stored to a file in the Memory Catalog and the Catalog of FIR Files is updated to show the new file.

To Create a User-Defined FIR Filter with the FIR Values Editor

In this procedure, you use the FIR Values editor to create and store an 8-symbol, windowed, sinc function filter with an oversample ratio of 4. The Oversample Ratio (OSR) is the number of filter coefficients per symbol.

You can define from 1 to 32 FIR coefficients, where the maximum combination of symbols and oversample ratio is 1024 coefficients.

The FIR Values editor allows a maximum filter length of 1024 coefficients, but the PSG hardware is limited to 512 symbols for arbitrary waveform generation. The number of symbols equals the number of coefficients divided by the oversample ratio. If you enter more than 512 symbols for arbitrary waveform generation, the PSG cannot use the filter; it will decimate the filter (throw away coefficients) until this condition is met. It will use the filter, but fine resolution may be lost from the impulse response.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Arb Waveform Generator** > **Digital Mod Define** > **Filter**.

3. Press **Define User FIR** > **More (1 of 2)**.

4. Press **Delete All Rows** > **Confirm Delete Of All Rows** > **More (2 of 2)**.

This brings up the FIR Values editor and clears the table of existing values.

Figure 5-3
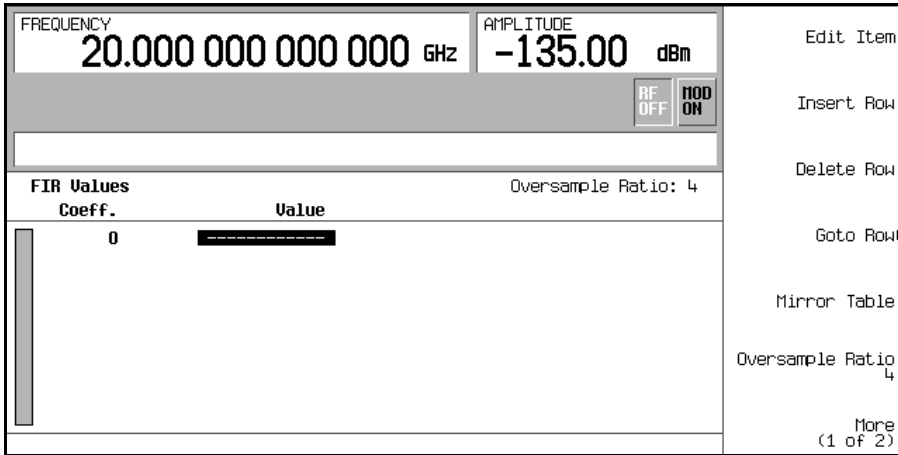
5. Press **Edit Item**. The Value field for coefficient 0 should be highlighted.

6. Use the numeric keypad to type the first value (−0.000076) from Table 5-1 and press **Enter**. As you press the numeric keys, the numbers are displayed in the active entry area. (If you make a mistake, you can correct it using the backspace key.)

7. Continue entering the coefficient values from the table until all 16 values have been entered:

Table 5-1

Coefficient	Value		Coefficient	Value
0	−0.000076		8	−0.035667
1	−0.001747		9	−0.116753
2	−0.005144		10	−0.157348
3	−0.004424		11	−0.088484
4	0.007745		12	0.123414
5	0.029610		13	0.442748
6	0.043940		14	0.767329
7	0.025852		15	0.972149

8. Press **Mirror Table**.

In a windowed sinc function filter, the second half of the coefficients are identical to the first half in reverse order. Since the signal generator provides a mirror table function that automatically duplicates the existing coefficient values in the reverse order, the last 16 coefficients (16 through 31) are automatically generated and the first of these coefficients (number 16) highlights, as shown in Figure 5-4.

Figure 5-4
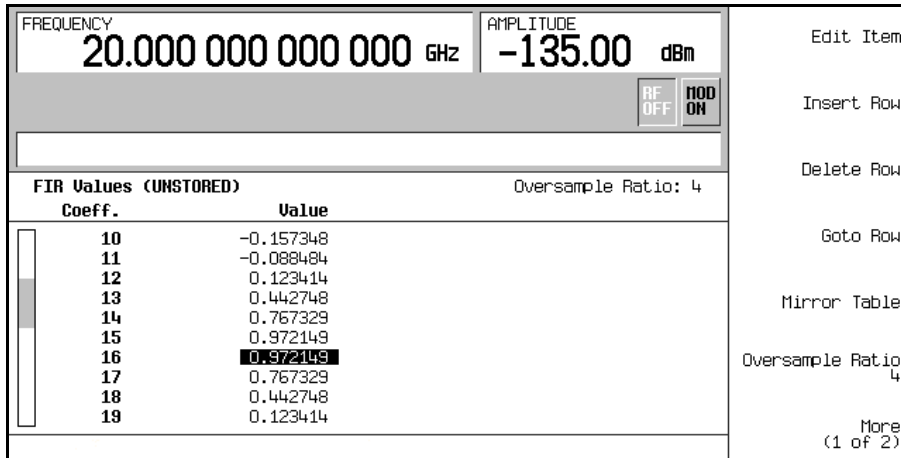
9. You can define from 1 to 32 FIR coefficients, where the maximum combination of symbols and oversample ratio is 1024 coefficients.

    The FIR Values editor allows a maximum filter length of 1024 coefficients, but the PSG hardware is limited to 512 symbols for arbitrary waveform generation. The number of symbols equals the number of coefficients divided by the oversample ratio. If you enter more than 512 symbols for arbitrary waveform generation, the PSG cannot use the filter; it will decimate the filter (throw away coefficients) until this condition is met. It will use the filter, but fine resolution may be lost from the impulse response.

    For this example, the desired OSR is 4, which is the default, so no action is necessary.

10. Press **More (1 of 2)** > **Display FFT** (fast Fourier transform).

    You will see a graph that shows the fast Fourier transform of the current set of FIR coefficients. The signal generator has the capability of graphically displaying the filter in both time and frequency dimensions.

Figure 5-5
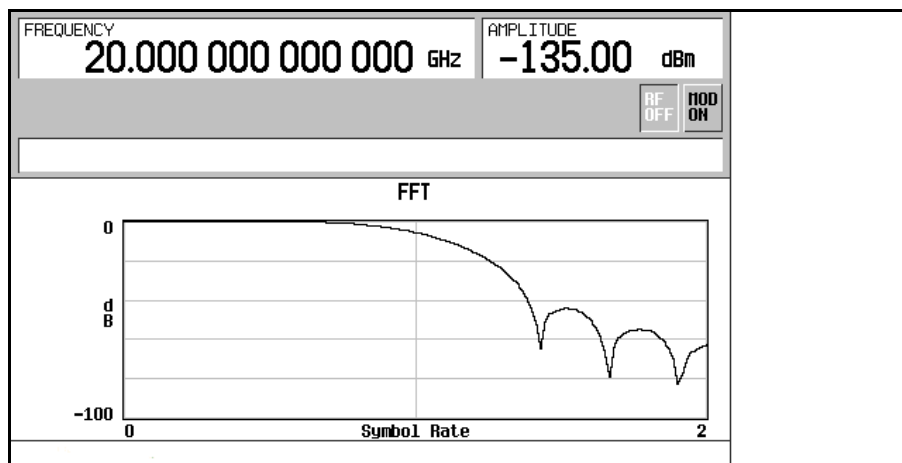
11. Press **Return**.

12. Press **Display Impulse Response**.

You should see a graph that shows the impulse response of the current set of FIR coefficients.

Figure 5-6

13. Press **Return**.

14. Press **Load/Store** > **Store To File**.

The Catalog of FIR Files appears along with the amount of memory available.

15. If there is already a file name occupying the active entry area, press the following keys:

Edit Keys > **Clear Text**

16. Using the alphabetic menu and the numeric keypad, enter NEWFIR1 as the file name.

17. Press **Enter.**

The NEWFIR1 file is the first file name listed. (If you have previously stored other FIR files, additional file names are listed below NEWFIR1.) The file type is FIR and the size of the file is 260 bytes. The amount of memory used is also displayed. The number of files that can be saved depends on the size of the files and the amount of memory used.

Figure 5-7

Working with Symbol Rates

In this section, you will learn about the following:

- Understanding Symbol Rates

- Using Symbol Rates

  — "To Set a Symbol Rate" on page 123

Understanding Symbol Rates

Symbol Rate allows you to access a menu from which you can set the rate at which I/Q symbols are fed to the I/Q modulator. The default transmission symbol rate can also be restored in this menu.

- Symbol Rate (displayed as `Sym Rate`) is the number of symbols per second that are transmitted using the modulation (displayed as `Mod Type`) along with the filter and filter alpha (displayed as `Filter`). Symbol rate directly influences the occupied signal bandwidth.

  Symbol Rate is the Bit Rate divided by the number of bits that can be transmitted with each symbol; this is also known as the Baud Rate.

- Bit Rate is the frequency of the system bit stream. The internal baseband generator (Option 002/602) automatically streams the selected Data Pattern at the appropriate rate to accommodate the symbol rate setting (Bit Rate = Symbols/s x Number of Bits/Symbol).

- Occupied Signal Bandwidth = Symbol Rate x (1 + Filter Alpha); therefore, the occupied signal bandwidth is dependent on the filter alpha of the Nyquist or Root Nyquist filter being used. (To change the filter alpha, refer to the procedure, "To Adjust the Filter Alpha of a Predefined Root Nyquist or Nyquist Filter" on page 112.)

Modulation Type		Bits Per Symbol	Bit Rate = Symbols/s x Number of Bits/Symbol	Internal Symbol Rate (Minimum Maximum)
PSK Phase Shift Keying	QPSK and OQPSK (quadrature phase shift keying and offset quadrature phase shift keying) which include: QPSK IS95 QPSK, Gray Coded QPSK, OQPSK, IS95 OQPSK	2	90 bps 100 Mbps	45 sps 50 Msps
	BPSK (binary phase shift keying)	1	45 bps 50 Mbps	45 sps 50 Msps
	π/4 DQPSK	2	90 bps 100 Mbps	45 sps 50 Msps
	8PSK (eight phase state shift keying)	3	135 bps 150 Mbps	45 sps 50 Msps
	16PSK (sixteen phase state shift keying)	4	180 sps 200 Mbps	45 sps 50 Msps
	D8PSK (eight phase state shift keying)	3	135 bps 150 Mbps	45 sps 50 Msps
MSK Minimum Shift Keying	MSK (GSM - Global System for Mobile Communications)	1	45 bps 50 Mbps	45 sps 50 Msps

Modulation Type		Bits Per Symbol	Bit Rate = Symbols/s x Number of Bits/Symbol	Internal Symbol Rate (Minimum Maximum)
FSK Frequency Shift Keying	2-Lvl FSK	1	45 bps 50 Mbps	45 sps 50 Msps
	4-Lvl FSK	2	90 bps 100 Mbps	45 sps 50 Msps
	8-Lvl FSK	3	135 bps 150 Mbps	45 sps 50 Msps
	16-Lvl FSK	4	180 bps 200 Mbps	45 sps 50 Msps
	C4FM	2	90 bps 100 Mbps	45 sps 50 Msps
QAM Quadrature Amplitude Modulation	4QAM	2	90 bps 100 Mbps	45 sps 50 Msps
	16QAM	4	180 bps 200 Mbps	45 sps 50 Msps
	32QAM	5	225 bps 250 Mbps	45 sps 50 Msps
	64QAM	6	270 bps 300 Mbps	45 sps 50 Msps
	128QAM (There is no preset value for this modulation, it must be user defined.)	7	315 bps 350 Mbps	45 sps 50 Msps
	256QAM	8	360 bps 400 Mbps	45 sps 50 Msps

To Set a Symbol Rate

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Symbol Rate**.

3. Enter a new symbol rate and press **Msps**, **ksps**, or **sps**.

Working with Modulation Types

Modulation Type is used to specify the type of modulation that will be applied to the carrier signal when the Mod On Off hardkey is set to On.

In addition, when the Custom Off On softkey is set to On, the BBG creates a sampled version of the I/Q waveform based on a random data pattern and the modulation type that has been selected.

In this section, you will learn about the following:

- Using a Predefined Modulation Type

To Select a Predefined PSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Modulation Type** > **Select** > **PSK**.

3. Press one of the following:

    BPSK, **π/4 DQPSK**, **8PSK**, **16PSK**, **D8PSK**
    or **QPSK** and **OQPSK** (if you select **QPSK and OQPSK**, press one of the following:
    QPSK, **IS95 QPSK**, **Gray Coded QPSK**, **OQPSK**, or **IS95 OQPSK**).

To Select a Predefined MSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Modulation Type** > **Select** > **MSK**.

To Select a Predefined FSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Modulation Type** > **Select** > **FSK**.

3. Press one of the following: **2-Lvl FSK**, **4-Lvl FSK**, **8-Lvl FSK**, **16-Lvl FSK**, **C4FM**, or **Freq Dev** (if you select **Freq Dev**, enter a new frequency deviation in Hertz.)

To Select a Predefined QAM Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **ARB Waveform Generator** > **Digital Mod Define** > **Modulation Type** > **Select** > **QAM**.

3. Press one of the following: **4QAM**, **16QAM**, **32QAM**, **64QAM**, **256QAM**

Working with Configuration of Hardware

In this section, you will learn about the following:

To Set a Delayed, Positive Polarity, External Single Trigger

Using this procedure, you learn how to utilize an external function generator to apply a delayed single-trigger to a custom multicarrier waveform.

1. Connect an Agilent 33120A function generator or equivalent to the signal generator PATT TRIGGER IN port, as shown in Figure 5-8.

Figure 5-8
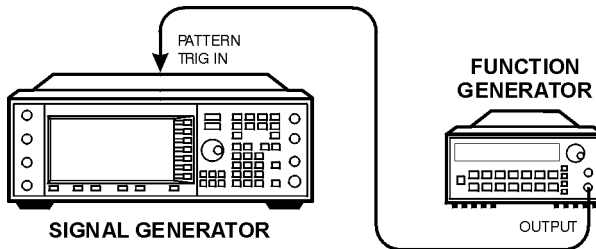
pk719b

2. On the signal generator, press **Preset**.

3. Press **Mode** > **Custom** > **Arb Waveform Generator**.

4. Press **Multicarrier Off On** until On is highlighted.

5. Press **Trigger** > **Single**.

6. Press **Trigger** > **Trigger Setup** >**Trigger Source** > **Ext**.

7. Press **Ext Polarity Neg Pos** until Pos is highlighted.

8. Press **Ext Delay Off On** until On is highlighted.

9. Press **Ext Delay Time** > **100** > **msec**.

The Custom Arb Waveform Generator has been configured to play a single multicarrier waveform 100 milliseconds after it detects a change in TTL state from low to high at the PATT TRIG IN rear panel connector.

10. Set the function generator waveform to a 0.1 Hz square wave at an output level of 0 to 5 V.

11. On the signal generator, press **Mode** > **Custom** > **Arb Waveform Generator** > **Digital Modulation Off On** until On is highlighted.

    This generates a waveform with the custom multicarrier state and the display changes to Dig Mod Setup: Multicarrier.

    During waveform generation, the DIGMOD and I/Q annunciators activate and the new custom multicarrier state is stored in volatile ARB memory. The waveform should be modulating the RF carrier.

12. Press **RF On/Off**.

    The externally single-triggered custom multicarrier waveform should be available at the signal generator's RF OUTPUT connector 100 ms after receiving a change in TTL state from low to high at the PATT TRIG IN.

To Set the ARB Reference to External or Internal

1. Press **Custom** > **Arb Waveform Generator** > **More (1 of 2)**.

2. Press **ARB Reference Ext Int** to select either external or internal as the waveform sample clock reference.

    • If you select Ext, you must enter the reference frequency (250 kHz to 100 MHz) and the reference signal must be applied to the BASEBAND GEN REF IN rear panel connector.

    • If you select Int, the internal clock is used for the arbitrary waveform (ARB) frequency reference.

To Set the External ARB Reference Frequency

The external Arb reference frequency is only used when the **ARB Reference Ext Int** softkey has been set to Ext (external).

1. Press **Custom** > Arb Waveform Generator > **More (1 of 2)**.

2. Press **Reference Freq**, enter a desired frequency (250 kHz to 100 MHz), and press **MHz**, **kHz**, or **Hz**.

6 Custom Real Time I/Q Baseband

This chapter describes the Custom Real Time I/Q Baseband mode which is available only in E8267C PSG vector signal generators.

This chapter includes the following major sections:

Overview of Using Custom Real Time I/Q Baseband Mode

Custom Real Time I/Q Baseband mode can produce a single carrier, but it can be modulated with real time data that allows real time control over all of the parameters that affect the signal. The single carrier signal that is produced can be modified by applying various data patterns, filters, symbol rates, modulation types, and burst shapes.

To begin using the Custom Real Time I/Q Baseband mode, start by selecting from a set of predefined modes (setups) or specify a setup by selecting a Data Pattern, Filter, Symbol Rate, Modulation Type, Burst Shape, Configure Hardware, Phase Polarity, and whether Diff Data Encode is off or on.

Working with Predefined Modes

To Select a Predefined Real Time Modulation Setup

When you select a predefined mode, default values for components of the setup (such as the data pattern, filter, symbol rate, modulation type, and the burst shape) are automatically specified.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband**.

3. Press **More (1 of 3)** > **More (2 of 3)** > **Predefined Mode** > **APCO 25 w/C4FM**.

4. Press **More (3 of 3)**.

   This selects a predefined mode where filtering, symbol rate, and modulation type are defined by the APCO 25 w/C4FM digital modulation standard and returns you to the top-level custom modulation menu.

To Deselect a Predefined Real Time Modulation Setup

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband**.

3. Press **More (1 of 3)** > **More (2 of 3)** > **Predefined Mode** > **None**.

4. Press **More (3 of 3)**.

   This deselects any predefined mode that has been previously selected.

Working with Data Patterns

In this section, you will learn about the following:

-

- Using a Predefined Data Pattern

  -

  -

  -

- Using a User-Defined Data Pattern

  User Files (user-defined data pattern files) can be created and modified using the signal generator's `Bit File Editor` or they can be created on a remote computer and moved to the signal generator for direct use; these remotely created data pattern files can also be modified with the Bit File Editor. For information on creating user-defined data files on a remote computer, see the programming guide.

  These procedures teaches you how to use the `Bit File Editor` to create, edit, and store user-defined data pattern files for use within the custom real-time I/Q baseband generator modulation. For this example, a user file is defined within a custom digital communication.

  -

  -

  -

  -

- Using Externally Supplied Data Patterns

  -

Understanding Data Patterns

Data allows you to select from predefined and user defined data patterns. Data Patterns are used for transmitting continuous streams of unframed data. When the Custom Off On softkey is set to On, the real-time custom I/Q symbol builder creates I/Q symbols based on the data pattern and modulation type that has been selected. Refer to "Working with Modulation Types" on page 153 to learn about selecting a modulation type.

You can select a data pattern from the following list:

- PN sequence allows you to access a menu (PN9, PN11, PN15, PN20, PN23) for internal data generation of pseudorandom sequences (pseudorandom noise sequences); a pseudorandom noise sequence is a periodic binary sequence approximating, in some sense, a Bernoulli "coin tossing" process with equiprobable outcomes.

- FIX4 0000 allows you to define a 4-bit repeating sequence data pattern and make it the active function. The selected 4-bit pattern will be repeated as necessary to provide a continuous stream of data.

- Other Patterns allows you to access a menu of choices (4 1's & 4 0's, 8 1's & 8 0's, 16 1's & 16 0's, 32 1's & 32 0's, or 64 1's & 64 0's) from which you can select a data pattern. Each pattern contains an equal number of ones and zeroes. The selected pattern will be repeated as necessary to provide a continuous stream of data.

- User File allows you to access a menu of choices from which you can create a file and store it to the Catalog of Bit Files, select from a Catalog of Bit Files and use it, or select from a Catalog of Bit Files, edit the file, and resave the file.

- Ext allows data patterns to be fed into the I/Q symbol builder, through the DATA port, in real-time.

To Select a Predefined PN Sequence Data Pattern

1. Press **Preset**.

2. Press **Mode > Custom > Real Time I/Q Baseband > Data > PN Sequence**.

3. Press one of the following: **PN9**, **PN11**, **PN15**, **PN20**, **PN23**.

To Select a Predefined Fixed 4-bit Data Pattern

1. Press **Preset**.

2. Press **Mode > Custom > Real Time I/Q Baseband > Data > FIX4**.

3. Press **1010 > Enter > Return**.

To Select a Predefined Data Pattern Containing an Equal Number of 1's & 0's

1. Press **Preset**.

2. Press **Mode > Custom > Real Time I/Q Baseband > Data > Other Patterns**.

3. Press one of the following:

   4 1's & 4 0's, **8 1's & 8 0's**, **16 1's & 16 0's**, **32 1's & 32 0's**, or **64 1's & 64 0's**.

To Create a Data Pattern User File with the Bit File Editor

In this procedure, you will use the Bit File Editor to create a Data Pattern User File and store the resultant file in the Memory Catalog; the Memory Catalog is a catalog of user files that has associated file management functions and a menu for choosing file types.

1. Press **Preset**.

2. Press **Mode** > Custom > **Real Time I/Q Baseband** > **Data** > **User File** > **Create File**.

   This opens the Bit File Editor which contains three columns: `Offset`, `Binary Data`, and `Hex Data`, as well as cursor position (`Position`), file size (`Size`), and file name (`Name`) indicators, as shown in the following figure.

Offset (in Hex)	**Bit Data**	**Cursor Position indicator (in Hex)**	**Hexadecimal Data**	**File Name indicator**
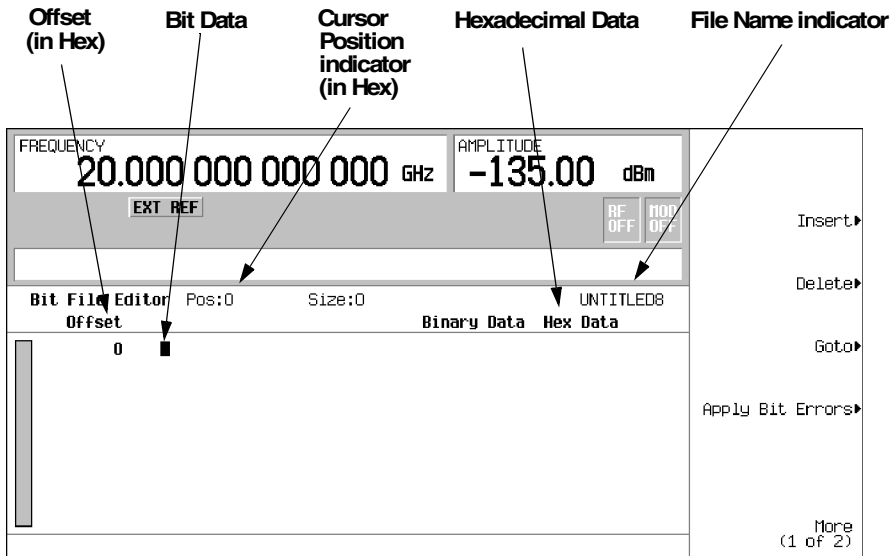
NOTE	When you create a new file, the default name appears as `UNTITLED`, or `UNTITLED1`, and so forth. This prevents overwriting previous files.

3. Enter the 32 bit values shown using the numeric keypad.

   Bit data is entered into the Bit File Editor in 1-bit format. The current hexadecimal value of the binary data is shown in the Hex Data column and the cursor position (in hexadecimal) is shown in the Position indicator.

Enter These Bit Values   **Cursor Position Indicator**   **Hexadecimal Data**

Offset	Binary Data	Hex Data
0	0110 1101 1011 0110 1110 1101 1011 0110	6DB6EDB6
20	1101 1011 0110 1101 1011 0110 1101 1011	DB6DB6DB
40	0111 0110 1101 0100 0110 1101 1011 0110	76D46DB6
60		

FREQUENCY 20.000 000 000 000 GHz   AMPLITUDE −135.00 dBm   EXT REF   RF OFF MOD OFF

Bit File Editor Pos:60   Size:96   UNTITLED8

Insert▶
Delete▶
Goto▶
Apply Bit Errors▶
More (1 of 2)

4. Press **More (1 of 2)** > **Rename** > **Editing Keys** > **Clear Text**.

5. Enter a file name (for example, USER1) using the alpha keys and the numeric keypad.

6. Press **Enter**.

   The user file should be renamed and stored to the Memory Catalog with the name USER1.

To Select a Data Pattern User File from the Catalog of Bit Files

In this procedure, you learn how to select a data pattern user file from the Catalog of Bit Files. If you have not created and stored a user-defined data file, complete the steps in the previous section, "To Create a Data Pattern User File with the Bit File Editor" on page 134.

1. Press **Preset**.

2. Press **Mode** > Custom > **Real Time I/Q Baseband** > **Data** > **User File**.

3. Highlight the file to be selected (for example, USER1).

4. Press **Edit File**.

   The Bit File Editor should open the selected file (for example, USER1).

To Modify an Existing Data Pattern User File

In this example, you learn how to modify an existing data pattern user file by navigating to a particular bit position and changing its value. Next, you will learn how to invert the bit values of an existing data pattern user file.

If you have not already created, stored, and recalled a data pattern user file, complete the steps in the previous sections, "To Create a Data Pattern User File with the Bit File Editor" on page 134 and "To Select a Data Pattern User File from the Catalog of Bit Files" on page 135.

Navigating the Bit Values of an Existing Data Pattern User File

1. Press **Goto** > **4** > **C** > **Enter**.

    This moves the cursor to bit position 4C, of the table, as shown in the following figure.
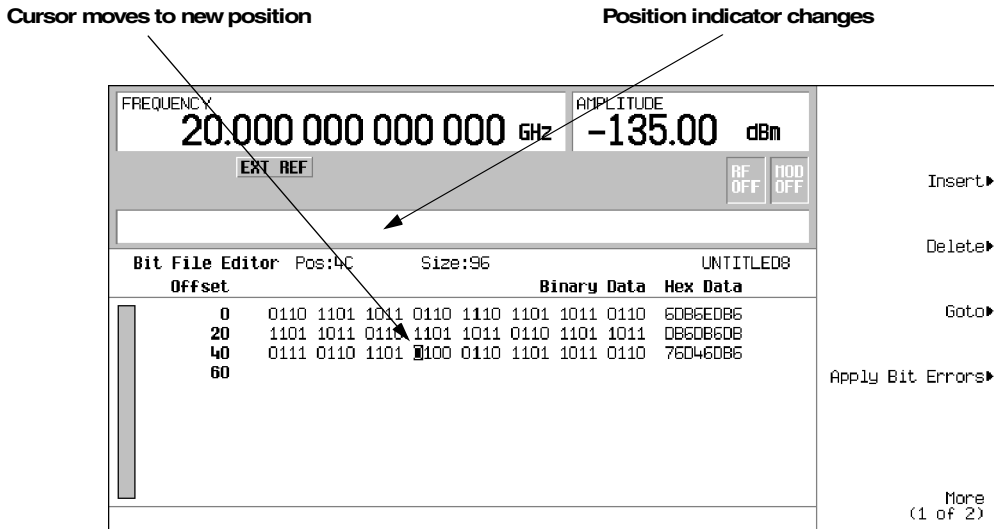
Inverting the Bit Values of an Existing Data Pattern User File

1. Press **1011**.

   This inverts the bit values that are positioned 4C through 4F. Notice that hex data in this row has now changed to 76DB6DB6, as shown in the following figure.

To Apply Bit Errors to an Existing Data Pattern User File

In this example, you learn how to apply bit errors to an existing data pattern user file. If you have not created and stored a data pattern user file, complete the steps in the previous section, "To Create a Data Pattern User File with the Bit File Editor" on page 134.

1. Press **Apply Bit Errors**.

2. Press **Bit Errors** > **5** > **Enter**.

3. Press **Apply Bit Errors**.

   Notice both **Bit Errors** softkeys change value as they are linked.

To Supply an External Real-Time Data Pattern

In this procedure, an external real time data pattern is supplied through DATA, DATA CLOCK, and SYMBOL SYNC connectors.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Data** > **Ext**.

3. **Connect your real-time data to the DATA input.**

4. **Connect your data clock trigger signal to DATA CLOCK input**.

5. **Connect your symbol sync trigger to the SYMBOL SYNC input**.

Working with Filters

In this section, you will learn about the following:

- "Understanding FIR Filters" on page 139

- Using a Predefined FIR Filter

    — "To Select a Predefined Root Nyquist, Nyquist, or Gaussian Filter" on page 141

    — "To Adjust the Filter Alpha of a Predefined Root Nyquist or Nyquist Filter" on page 141

    — "To Adjust the Bandwidth-Bit-Time (BbT) Product of a Predefined Gaussian Filter" on page 141

    — "To Optimize the FIR Filter for EVM or ACP" on page 141

    — "To Select a Predefined Rectangle Filter" on page 141

    — "To Select an APCO 25-Specified C4FM Filter" on page 141

    — "To Restore Default FIR Filter Parameters" on page 142

- Using a User-Defined FIR Filter

    FIR filters can be created and modified by defining the FIR coefficients or by defining the oversample ratio (number of filter coefficients per symbol) to be applied to your own custom FIR filter.

    — "To Modify Predefined FIR Coefficients for a Gaussian Filter with the FIR Values Editor" on page 142

    — "To Create a User-Defined FIR Filter with the FIR Values Editor" on page 144

Understanding FIR Filters

Filter allows you to select the type of filter to be applied to the signal being generated, define your own Finite Impulse Response (FIR) filter parameters, change the filter alpha being used for Root Nyquist or Nyquist, change the BbT for Gaussian, optimize your FIR filter for Error Vector Magnitude (EVM) or Adjacent Channel Power (ACP), and restore all of the filter parameters back to their original default state.

NOTE	These procedures only work for FIR filters created within Custom Real Time I/Q Baseband mode and will not work with downloaded user files such as Matlab files.

- Select (predefined filters):

  — Root Nyquist selects a root-raised cosine pre-modulation FIR filter. Root Nyquist filters can be used when you want to place half of the filtering in the transmitter and the other half of the filtering in the receiver. The ideal root-raised cosine filter frequency response consists of unity gain at low frequencies, the square root of raised cosine function in the middle, and total attenuation at high frequencies. The width of the middle frequencies are defined by the roll off factor or Filter Alpha (0 < Filter Alpha < 1).

  — Nyquist selects a raised cosine pre-modulation FIR filter. Nyquist filters can be used to reduce the amount of bandwidth required by the signal be produced without loosing information. The ideal raised cosine filter frequency response consists of unity gain at low frequencies, a raised cosine function in the middle, and total attenuation at high frequencies. The width of the middle frequencies are defined by the roll off factor or Filter Alpha (0 < Filter Alpha < 1).

  — Gaussian selects a Gaussian pre-modulation FIR filter.

  — User FIR allows you to select a FIR filter from a Catalog of FIR filters; this selection is used if the predefined FIR filters (Root Nyquist, Nyquist, Gaussian, etc.) do not meet your filtering needs. For further information, refer to the Define User FIR softkey.

  — Rectangle selects a rectangular pre-modulation FIR filter.

  — APCO 25 C4FM selects an APCO 25-specified C4FM filter; this is a Nyquist filter with an alpha of 0.200 which is combined with a shaping filter.

- Define User FIR is used if the predefined FIR filters do not meet your filtering needs. You can define your own FIR coefficients for a FIR filter and set the oversample ratio (number of filter coefficients per symbol) to be applied to a your own custom FIR filter.

- Filter Alpha allows you to adjust the filter alpha when Nyquist or root Nyquist filters are selected. This feature applies only to Root Nyquist and Nyquist filters. If a Gaussian filter is in use, you will see Filter BbT; the softkey is grayed out when any other filter is selected.

- Optimize FIR for EVM ACP allows you to optimize the FIR filter being used for minimized error vector magnitude (EVM) or for minimized adjacent channel power (ACP). This feature applies only to Nyquist and root Nyquist filters; the softkey is grayed out when any other filter is selected.

- Restore Default Filters allows you to replace the current FIR filter with the default FIR filter for the selected format.

To Select a Predefined Root Nyquist, Nyquist, or Gaussian Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Select** > and select one of the following: Root Nyquist | Nyquist | Gaussian.

To Adjust the Filter Alpha of a Predefined Root Nyquist or Nyquist Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Filter Alpha**.

3. Enter a new Filter Alpha value and press **Enter**.

To Adjust the Bandwidth-Bit-Time (BbT) Product of a Predefined Gaussian Filter

1. Press **Filter** > **Select** > **Gaussian**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Select** > **Gaussian**.

3. Press **Filter BbT**.

4. Enter a new Bandwidth-Bit-Time (BbT) product filter parameter and press **Enter**.

To Optimize the FIR Filter for EVM or ACP

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Optimize FIR For EVM or ACP**.

   The FIR filter is then optimized for minimum error vector magnitude (EVM) or for minimum adjacent channel power (ACP). This feature applies only to Nyquist and root Nyquist filters; the softkey is grayed out when any other filter is selected.

To Select a Predefined Rectangle Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Select** > **More (1 of 2)** > **Rectangle**.

To Select an APCO 25-Specified C4FM Filter

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Select** > **(More 1 of 2)** > **APCO 25 C4FM**.

   This selects a Nyquist filter with an alpha of 0.200 which is combined with a shaping filter.

To Restore Default FIR Filter Parameters

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter** > **Restore Default Filter**.

   This replaces the current FIR filter with the default filter for the selected modulation format.

To Modify Predefined FIR Coefficients for a Gaussian Filter with the FIR Values Editor

You can define from 1 to 32 FIR coefficients, where the maximum combination of symbols and oversample ratio is 1024 coefficients.

The FIR Values editor allows a maximum filter length of 1024 coefficients, but the PSG hardware is limited to 64 symbols for real-time and 512 symbols for arbitrary waveform generation. The number of symbols equals the number of coefficients divided by the oversample ratio. If you enter more than 64 symbols for real-time and 512 symbols for arbitrary waveform generation, the PSG cannot use the filter.

FIR filters stored in signal generator memory can easily be modified using the FIR Values editor. In this example, you will load the FIR Values editor with coefficient values from a default FIR filter (or, if one has been defined, a user-defined FIR file that has been stored in the Memory Catalog), modify the coefficient values, and store the new file to the Memory Catalog.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter**.

3. Press **Define User FIR** > **More (1 of 2)** > **Load Default FIR** > **Gaussian**.

4. Press **Filter BbT** > **0.300** > **Enter**.

5. Press **Filter Symbols** > **8** > **Enter**.

6. Press **Generate**.

NOTE   The actual oversample ratio during modulation is automatically selected by the instrument. A value between 4 and 16 is chosen dependent on the symbol rate, the number of bits per symbol of the modulation type, and the number of symbols.

---

7. Press **Display Impulse Response**.

   You should see a graph that shows the impulse response of the current set of FIR coefficients.

Figure 6-1

8. Press **Return**.

9. Highlight coefficient 15.

10. Press **0 > Enter**.

11. Press **Display Impulse Response**.

Figure 6-2

The graphic display can provide a useful troubleshooting tool (in this case, it indicates that a coefficient value is set incorrectly, resulting in an improper Gaussian response).

12. Press **Return**.

13. Highlight coefficient 15.

14. Press **1** > **Enter**.

15. Press **Load/Store** > **Store To File**.

16. Name the file NEWFIR2.

17. Press **Enter**.

The contents of the current FIR Values editor are stored to a file in the Memory Catalog and the Catalog of FIR Files is updated to show the new file.

To Create a User-Defined FIR Filter with the FIR Values Editor

In this procedure, you use the FIR Values editor to create and store an 8-symbol, windowed, sinc function filter with an oversample ratio of 4.

You can define from 1 to 32 FIR coefficients, where the maximum combination of symbols and oversample ratio is 1024 coefficients.

The FIR Values editor allows a maximum filter length of 1024 coefficients, but the PSG hardware is limited to 64 symbols for real-time and 512 symbols for arbitrary waveform generation. The number of symbols equals the number of coefficients divided by the oversample ratio. If you enter more than 64 symbols for real-time and 512 symbols for arbitrary waveform generation, the PSG cannot use the filter.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Filter**.

3. Press **Define User FIR** > **More (1 of 2)**.

4. Press **Delete All Rows** > **Confirm Delete Of All Rows** > **More (2 of 2)**.

This brings up the FIR Values editor and clears the table of existing values.

Figure 6-3

5. Press **Edit Item**.

   The Value field for coefficient 0 should be highlighted.

6. Use the numeric keypad to type the first value (−0.000076) from Table 6-1 and press **Enter**. As you press the numeric keys, the numbers are displayed in the active entry area. (If you make a mistake, you can correct it using the backspace key.)

7. Continue entering the coefficient values from the table until all 16 values have been entered:

   Table 6-1

Coefficient	Value		Coefficient	Value
0	−0.000076		8	−0.035667
1	−0.001747		9	−0.116753
2	−0.005144		10	−0.157348
3	−0.004424		11	−0.088484
4	0.007745		12	0.123414
5	0.029610		13	0.442748
6	0.043940		14	0.767329
7	0.025852		15	0.972149

8. Press **Mirror Table**.

   In a windowed sinc function filter, the second half of the coefficients are identical to the first half in
   reverse order. Since the signal generator provides a mirror table function that automatically duplicates
   the existing coefficient values in the reverse order, the last 16 coefficients (16 through 31) are
   automatically generated and the first of these coefficients (number 16) highlights, as shown in
   Figure 6-4.

   Figure 6-4
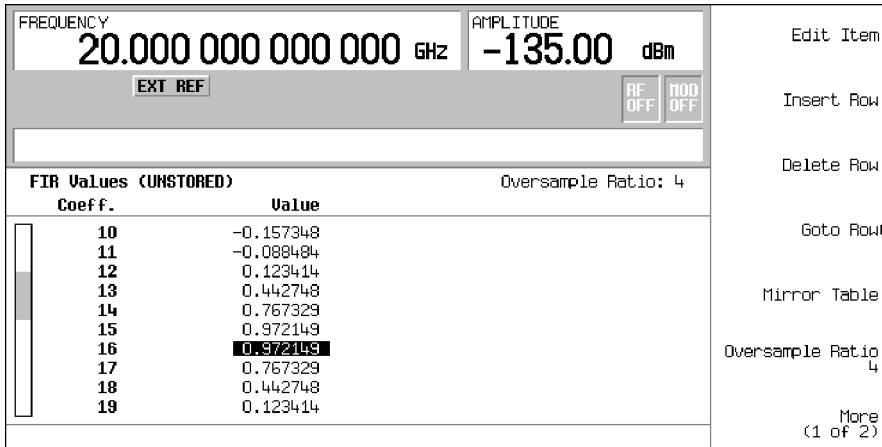
9. For this example, the desired OSR is 4, which is the default, so no action is necessary.

   The Oversample Ratio (OSR) is the number of filter coefficients per symbol. Acceptable values range
   from 1 through 32; the maximum combination of symbols and oversampling ratio allowed by the FIR
   Values editor is 1024. The instrument hardware, however, is actually limited to 32 symbols, an
   oversample ratio between 4 and 16, and 512 coefficients. So if you enter more than 32 symbols or 512
   coefficients, the instrument is unable to use the filter. If the oversample ratio is different from the
   internal, optimally selected one, then the filter is automatically resampled to an optimal oversample
   ratio.

10. Press **More (1 of 2)** > **Display FFT** (fast Fourier transform).

    You will see a graph that shows the fast Fourier transform of the current set of FIR coefficients. The
    signal generator has the capability of graphically displaying the filter in both time and frequency
    dimensions.

Figure 6-5
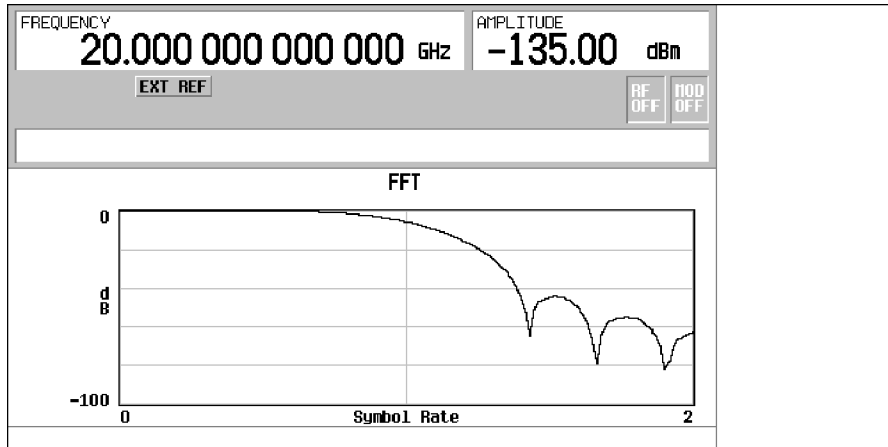
11. Press **Return**.

12. Press **Display Impulse Response**.

    You should see a graph that shows the impulse response of the current set of FIR coefficients.

Figure 6-6

13. Press **Return**.

14. Press **Load/Store > Store To File**.

    The Catalog of FIR Files appears along with the amount of memory available.

15. If there is already a file name occupying the active entry area, press the following keys:

Edit Keys > **Clear Text**

16. Using the alphabetic menu and the numeric keypad, enter NEWFIR1 as the file name.

17. Press **Enter.**

The NEWFIR1 file is the first file name listed. (If you have previously stored other FIR files, additional file names are listed below NEWFIR1.) The file type is FIR and the size of the file is 260 bytes. The amount of memory used is also displayed. The number of files that can be saved depends on the size of the files and the amount of memory used.

Figure 6-7

Working with Symbol Rates

In this section, you will learn about the following:

- Understanding Symbol Rates

- Using Symbol Rates

  —

  —

Understanding Symbol Rates

Symbol Rate allows you to access a menu from which you can set the rate at which I/Q symbols are fed to the I/Q modulator. The default transmission symbol rate can also be restored in this menu.

- Symbol Rate (displayed as `Sym Rate`) is the number of symbols per second that are transmitted using the modulation (displayed as `Mod Type`) along with the filter and filter alpha (displayed as `Filter`). Symbol rate directly influences the occupied signal bandwidth.

  Symbol Rate is the Bit Rate divided by the number of bits that can be transmitted with each symbol; this is also known as the Baud Rate.

- Bit Rate is the frequency of the system bit stream. The internal baseband generator (Option 002/602) automatically streams the selected Data Pattern at the appropriate rate to accommodate the symbol rate setting (Bit Rate = Symbols/s x Number of Bits/Symbol).

- Occupied Signal Bandwidth = Symbol Rate x (1 + Filter Alpha); therefore, the occupied signal bandwidth is dependent on the filter alpha of the Nyquist or Root Nyquist filter being used. (To change the filter alpha, refer to the procedure, .)

Modulation Type		Bits Per Symbol	Bit Rate = Symbols/s x Number of Bits/Symbol	Internal Symbol Rate (Minimum Maximum)	External Symbol Rate (Minimum Maximum)
PSK Phase Shift Keying	QPSK and OQPSK (quadrature phase shift keying and offset quadrature phase shift keying) which include: QPSK IS95 QPSK, Gray Coded QPSK, OQPSK, IS95 OQPSK	2	90 bps 100 Mbps	45 sps 50 Msps	45 sps 25 Msps
	BPSK (binary phase shift keying)	1	45 bps 50 Mbps	45 sps 50 Msps	45 sps 50 Msps
	$\pi/4$ DQPSK	2	90 bps 100 Mbps	45 sps 50 Msps	45 sps 25 Msps
	8PSK (eight phase state shift keying)	3	135 bps 150 Mbps	45 sps 50 Msps	45 sps 16.67 Msps
	16PSK (sixteen phase state shift keying)	4	180 sps 200 Mbps	45 sps 50 Msps	45 sps 12.5 Msps
	D8PSK (eight phase state shift keying)	3	135 bps 150 Mbps	45 sps 50 Msps	45 sps 16.67 Msps
MSK Minimum Shift Keying	MSK (GSM - Global System for Mobile Communications)	1	45 bps 50 Mbps	45 sps 50 Msps	45 sps 50 Msps

Modulation Type		Bits Per Symbol	Bit Rate = Symbols/s x Number of Bits/Symbol	Internal Symbol Rate (Minimum Maximum)	External Symbol Rate (Minimum Maximum)
FSK Frequency Shift Keying	2-Lvl FSK	1	45 bps 50 Mbps	45 sps 50 Msps	45 sps 50 Msps
	4-Lvl FSK	2	90 bps 100 Mbps	45 sps 50 Msps	45 sps 25 Msps
	8-Lvl FSK	3	135 bps 150 Mbps	45 sps 50 Msps	45 sps 16.67 Msps
	16-Lvl FSK	4	180 bps 200 Mbps	45 sps 50 Msps	45 sps 12.5 Msps
	C4FM	2	90 bps 100 Mbps	45 sps 50 Msps	45 sps 25 Msps
QAM Quadrature Amplitude Modulation	4QAM	2	90 bps 100 Mbps	45 sps 50 Msps	45 sps 25 Msps
	16QAM	4	180 bps 200 Mbps	45 sps 50 Msps	45 sps 12.5 Msps
	32QAM	5	225 bps 250 Mbps	45 sps 50 Msps	45 sps 10 Msps
	64QAM	6	270 bps 300 Mbps	45 sps 50 Msps	45 sps 8.33 Msps
	128QAM (There is no preset value for this modulation, it must be user defined.)	7	315 bps 350 Mbps	45 sps 50 Msps	45 sps 7.14 Msps
	256QAM	8	360 bps 400 Mbps	45 sps 50 Msps	45 sps 6.25 Msps

To Set a Symbol Rate

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Symbol Rate**.

3. Enter a new symbol rate and press **Msps**, **ksps**, or **sps**.

To Restore the Default Symbol Rate

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Symbol Rate** > **Restore Default Symbol Rate**.

   This replaces the current symbol rate with the default symbol rate for the selected modulation format.

Working with Modulation Types

In this section, you will learn about the following:

Understanding Modulation Types

Modulation Type is used to specify the type of modulation that will be applied to the carrier signal when the Mod On Off hardkey is set to On.

In addition, when the Custom Off On softkey is set to On, the real-time custom I/Q symbol builder creates I/Q symbols based on the data pattern and modulation type that has been selected. Refer to "Working with Data Patterns" on page 132 to learn about selecting a data pattern.

You can select a modulation type from the following list:

- Select allows you to access a menu from which you can select predefined modulations (PSK, MSK, FSK, QAM) or user-defined modulation types (I/Q and FSK) that have been previously defined and saved in the Memory Catalog.

- Define User I/Q allows you to create user-defined I/Q modulation types that can be used immediately or saved to the Memory Catalog for reuse. Once these user-defined I/Q modulation types have been defined and saved, they are available through the Select menu.

- Define User FSK allows you to create user-defined FSK modulation types that can be used immediately or saved to the Memory Catalog for reuse. Once these user-defined FSK modulation types have been defined and saved, they are available through the Select menu.

- Restore Default Modulation Type allows you to restore all of the modulation parameters back to their original default state.

To Select a Predefined PSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Select** > **PSK**.

3. Press one of the following: **BPSK**, **π/4 DQPSK**, **8PSK**, **16PSK**, **D8PSK** or **QPSK** and **OQPSK** (if you select QPSK and OQPSK, press one of the following: **QPSK**, **IS95 QPSK**, **Gray Coded QPSK**, **OQPSK**, or **IS95 OQPSK**).

To Select a Predefined MSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Select** > **MSK** > **Phase Dev**.

3. Enter a new phase deviation angle and press **deg**.

To Select a Predefined FSK Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Select** > **FSK**.

3. Press one of the following: **2-Lvl FSK**, **4-Lvl FSK**, **8-Lvl FSK**, **16-Lvl FSK**, **C4FM**, or **Freq Dev** (if you select **Freq Dev**, enter a new frequency deviation in Hertz.)

To Select a Predefined QAM Modulation Type

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Select** > **QAM**.

3. Press one of the following: **4QAM**, **16QAM**, **32QAM**, **64QAM**, **256QAM**

To Create a 128QAM I/Q Modulation Type User File with the I/Q Values Editor

In I/Q modulation schemes, symbols appear in default positions in the I/Q plane. Using the `I/Q Values` editor, you can define your own symbol map by changing the position of one or more symbols.

Use the following procedure to create and store a 128-symbol QAM modulation.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User I/Q** > **More (1 of 2)** > **Load Default I/Q Map** > **QAM** > **256QAM**.

   This loads a default 256QAM I/Q modulation into the `I/Q Values` editor.

3. Press **More (2 of 2)** > **Display I/Q Map**.

   Figure 6-8
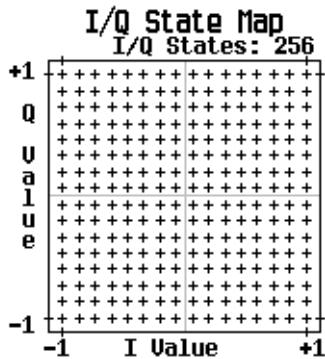
   In the next steps, you will delete specific portions of this I/Q constellation and change it into a 128QAM with 128 I/Q states.

NOTE	Although this procedure shows you a quick way to implement a 128QAM modulation format, it has a slight drawback.
	This approach does not take full advantage of the I/Q modulator's dynamic range. This occurs because, in this procedure, all of the deleted points are simply deleted from a 256QAM constellation. The remaining points that make up the 128QAM constellation are then all that is left; the points that are left are closer together than if you were to map out each point specifically.
	In addition, this approach does not allow you to define the bit pattern associated with each symbol point. To do this, the 128QAM constellation must be defined one point at a time.

4. Press **Return** > **Goto Row** > **0011 0000** > **Enter**; this is row 48.

5. Press the **Delete Row** softkey 16 times.

   Repeat this pattern of steps while using the following table:

Goto Row...	Press the **Delete Row** softkey...
0110 0000 (96)	16 times
1001 0000 (144)	16 times
1100 0000 (192)	16 times
0001 0000 (16)	4 times
0001 0100 (20)	4 times
0001 1000 (24)	8 times
0011 0000 (48)	4 times
0011 0100 (52)	4 times
0011 1000 (56)	4 times
0101 1000 (88)	8 times
0111 0000 (112)	4 times
0111 0100 (116)	4 times
0111 1000 (120)	8 times

6. Press **Display I/Q Map** to view the new constellation that has been created. The I/Q State Map in this example has 128 symbols.

Figure 6-9
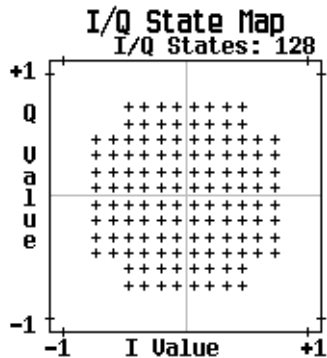
7. Press **Return**.

   When the contents of an `I/Q Values` table have not been stored, `I/Q Values (UNSTORED)` appears on the display.

8. Press **More (1 of 2)** > **Load/Store** > **Store To File**.

   If there is already a file name from the `Catalog of IQ Files` occupying the active entry area, press the following keys:

   Editing Keys > **Clear Text**

9. Enter a file name (for example, `128QAM`) using the alpha keys and the numeric keypad.

10. Press **Enter**.

    The user-defined I/Q State Map should now be stored in the `Catalog of IQ Files`.

To Create a QPSK I/Q Modulation Type User File with the I/Q Values Editor

In I/Q modulation schemes, symbols appear in default positions in the I/Q plane. Using the `I/Q Values` editor, you can define your own symbol map by changing the position of one or more symbols.

Use the following procedure to create and store a 4-symbol unbalanced QPSK modulation.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User I/Q** > **More (1 of 2)** > **Delete All Rows** > **Confirm Delete All Rows**.

   This loads a default 4QAM I/Q modulation and clears the `I/Q Values` editor.

3. Enter the I and Q values listed in the following table:

Symbol	Data	I Value	Q Value
0	0000	0.500000	1.000000
1	0001	−0.500000	1.000000
2	0010	0.500000	−1.000000
3	0011	−0.500000	−1.000000

   a. Press **0.5** > **Enter**.

   b. Press **1** > **Enter**.

   c. Enter the remaining I and Q values.

   As the I value updates, the highlight moves to the first Q entry (and provides a default value of 0) and an empty row of data appears below the first row. As the Q value updates, the highlight moves to the next I value. As you press the numeric keys, the numbers display in the active entry area. If you make a mistake, use the backspace key and retype.

   Also note that `0.000000` appears as the first entry in the list of `Distinct Values`, and that `0.500000` and `1.000000` are listed as the distinct values.

4. Press **More (2 of 2)** > **Display I/Q Map**.

   An I/Q State Map is displayed from the current values in the `I/Q Values` table.

   The I/Q State Map in this example has four symbols. The I/Q State Map uses the following *four* unique values: 0.5, 1.0, −0.5, and −1.0 to create the four symbols. It is not the number of values that defines how many symbols a map has, but how those values are combined.

5. Press **Return**.

   When the contents of an `I/Q Values` table have not been stored, `I/Q Values (UNSTORED)` appears on the display.

6. Press **More (1 of 2)** > **Load/Store** > **Store To File**.

   If there is already a file name from the `Catalog of IQ Files` occupying the active entry area, press the following keys:

   Editing Keys > **Clear Text**

7. Enter a file name (for example, `NEW4QAM`) using the alpha keys and the numeric keypad.

8. Press **Enter**.

   The user-defined I/Q State Map should now be stored in the `Catalog of IQ Files` and can be recalled even after the E8267C PSG signal generator has been turned off.

To Modify a Predefined I/Q Modulation Type (I/Q Symbols) and Simulate Magnitude Errors and Phase Errors

Use the following procedure to manipulate symbol locations which simulate magnitude and phase errors. In this example, you edit a 4QAM constellation to move one symbol closer to the origin.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User I/Q** > **More (1 of 2)** > **Load Default I/Q Map** > **QAM** > **4QAM**.

   This loads a default 4QAM I/Q modulation into the I/Q Values editor.

3. Press **More (2 of 2)**.

4. In the I/Q Values editor, navigate to Data 00000000 and press **Edit Item**.

5. Press **.235702** > **Enter**.

6. Press **.235702** > **Enter**.

   As you enter the numbers using the numeric keypad, they are displayed in the active entry area. If you make a mistake, use the backspace key and retype. The I value updates and the highlight moves to the first Q entry. Next, the Q value updates and the highlight moves to the following I entry.

7. Press **Display I/Q Map**.

   Note that one symbol has moved, as shown.

   Figure 6-10
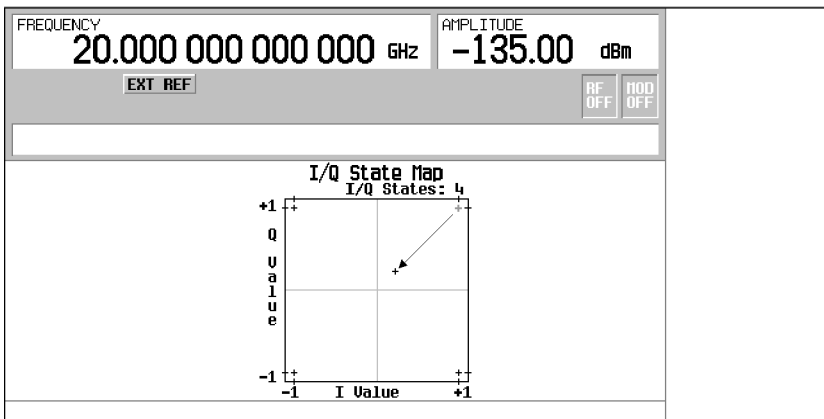
To Create an FSK Modulation Type User File with the Frequency Values Editor

During this procedure, you will set the frequency deviation for data `00`, `01`, `10`, and `11` to configure a user-defined FSK modulation.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User FSK** > **More (1 of 2)** > **Delete All Rows** > **Confirm Delete All Rows**.

   This accesses the `Frequency Values` editor and clears the previous values.

3. Press **600** > **Hz**.

4. Press **1.8** > **kHz**.

5. Press **-600** > **Hz**.

6. Press **-1.8** > **kHz**.

   Each time you enter a value, the `Data` column increments to the next binary number, up to a total of 16 data values (from `0000` to `1111`). An unstored file of frequency deviation values is created for your custom 4-level FSK file.

7. Press **Load/Store** > **Store To File**.

   If there is already a file name from the `Catalog of FSK Files` occupying the active entry area, press the following keys:

   Edit Keys > **Clear Text**

8. Enter a file name (for example, `NEWFSK`) using the alpha keys and the numeric keypad.

9. Press **Enter**.

   The user-defined FSK modulation should now be stored in the `Catalog of FSK Files`.

To Modify a Predefined FSK Modulation Type User File with the Frequency Values Editor

Using the `Frequency Values` editor, you can define, modify, and store user-defined frequency shift keying modulation.

The `Frequency Values` editor is available for custom Real-Time I/Q Baseband mode, but is not available for waveforms generated in custom Arb Waveform Generator mode.

In this example, you learn how to add errors to a default FSK modulation.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User FSK** > **More (1 of 2)** > **Load Default FSK**.

3. Press **Freq Dev** > **1.8** > **kHz**.

4. Press **4-Lvl FSK**.

   This sets the frequency deviation and opens the `Frequency Values` editor with the 4-level FSK default values displayed. The frequency value for data `0000` is highlighted.

5. Press **-1.81** > **kHz**.

6. Press **-590** > **Hz**.

7. Press **1.805** > **kHz**.

8. Press **610** > **Hz**.

   As you modify the frequency deviation values, the cursor moves to the next data row. An unstored file of frequency deviation values is created for your custom 4-level FSK file.

9. Press **Load/Store** > **Store To File**.

   If there is already a file name from the `Catalog of FSK Files` occupying the active entry area, press the following keys:

   Edit Keys > **Clear Text**

10. Enter a file name (for example, `NEWFSK`) using the alpha keys and the numeric keypad.

11. Press **Enter**.

   The user-defined FSK modulation should now be stored in the `Catalog of FSK Files`.

Working with Burst Shapes

In this section, you will learn about the following:

- "Understanding Burst Shapes" on page 163
- "Configuring the Burst Rise and Fall Parameters" on page 165
- "To Create and Store User-Defined Burst Shape Curves" on page 165

You can adjust the shape of the rise time curve and the fall time curve using the Rise Shape and Fall Shape editors. Each editor allows you to enter up to 256 values, equidistant in time, to define the shape of the curve. The values are then resampled to create the cubic spline that passes through all of the sample points.

The Rise Shape and Fall Shape editors are available for custom real-time I/Q baseband generator waveforms. They are not available for waveforms generated by the dual arbitrary waveform generator.

You can also design burst shape files externally and download the data to the signal generator. For more information, see the programming guide.

Understanding Burst Shapes

Burst Shape allows you to access a menu from which you can modify the rise and fall time, rise and fall delay, and the burst shape (either sine or user file defined). In addition, you can define the shape of the burst and preview the burst shape through a Rise Shape Editor, or restore all of the burst shape parameters back to their original default state.

Rise time	the period of time, specified in bits, where the burst increases from a minimum of −70 dB (0) to full power (1).
Fall time	the period of time, specified in bits, where the burst decreases from full power (1) to a minimum of −70 dB (0).
Rise delay	the period of time, specified in bits, that the start of the burst rise is delayed. Rise delay can be either negative or positive. Entering a delay other than zero shifts the full power point earlier or later than the beginning of the first useful symbol.
Fall delay	the period of time, specified in bits, that the start of the burst fall is delayed. Fall delay can be either negative or positive. Entering a delay other than zero shifts the full power point earlier or later than the end of the last useful symbol.

User-defined burst shape

up to 256 user-entered values which define the shape of the curve in the specified rise or fall time. The values can vary between 0 (no power) and 1 (full power) and are scaled linearly. Once specified, the values are resampled as necessary to create the cubic spline that passes through all of the sample points.

The default burst shape of each format is implemented according to the standards of the format selected. You can, however, modify the following aspects of the burst shape:
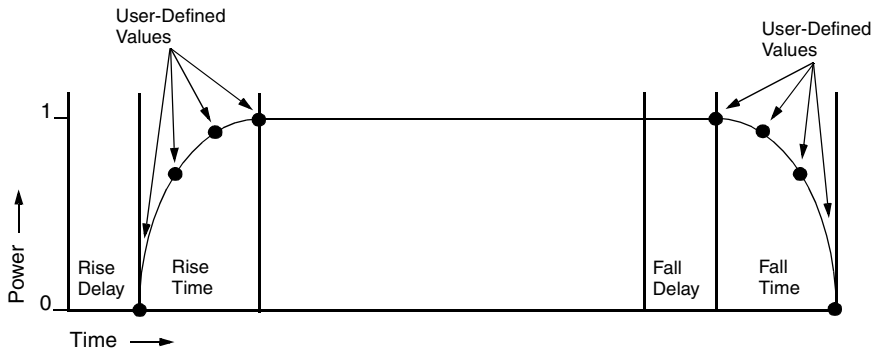
Burst shape maximum rise and fall time values are affected by the following factors:

- the symbol rate

- the modulation type

When the rise and fall delays equal 0, the burst shape attempts to synchronize the maximum burst shape power to the beginning of the first valid symbol and the ending of the last valid symbol.

If you find that the error vector magnitude (EVM) or adjacent channel power (ACP) increases when you turn bursting on, you can adjust the burst shape to assist with troubleshooting.

Configuring the Burst Rise and Fall Parameters

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Burst Shape**.

3. Press **Rise Time** > **5** > **bits**.

4. Press **Rise Delay** > **1** > **bits**.

5. Press **Fall Time** > **5** > **bits**.

6. Press **Fall Delay** > **1** > **bits**.

This configures the burst shape for the custom real-time I/Q baseband digital modulation format. For instructions on creating and applying user-defined burst shape curves, see "To Create and Store User-Defined Burst Shape Curves" on page 165.

To Create and Store User-Defined Burst Shape Curves

Using this procedure, you learn how to enter rise shape sample values and mirror them as fall shape values to create a symmetrical burst curve.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Burst Shape**.

3. Press **Define User Burst Shape** > **More (1 of 2)** > **Delete All Rows** > **Confirm Delete Of All Rows**.

4. Enter values similar to the sample values in the following table:

Rise Shape Editor			
Sample	**Value**	**Sample**	**Value**
0	0.000000	5	0.900000
1	0.400000	6	0.950000
2	0.600000	7	0.980000
3	0.750000	8	0.990000
4	0.830000	9	1.000000

   a. Highlight the value (1.000000) for sample 1.

   b. Press **.4** > **Enter**.

   c. Press **.6** > **Enter**.

   d. Enter the remaining values for samples 3 through 9 from the table above.

e. Press **More (2 of 2)** > **Edit Fall Shape** > **Load Mirror Image of Rise Shape** >
    Confirm Load Mirror Image of Rise Shape.

This changes the fall shape values to a mirror image of the rise shape values.

Figure 6-11
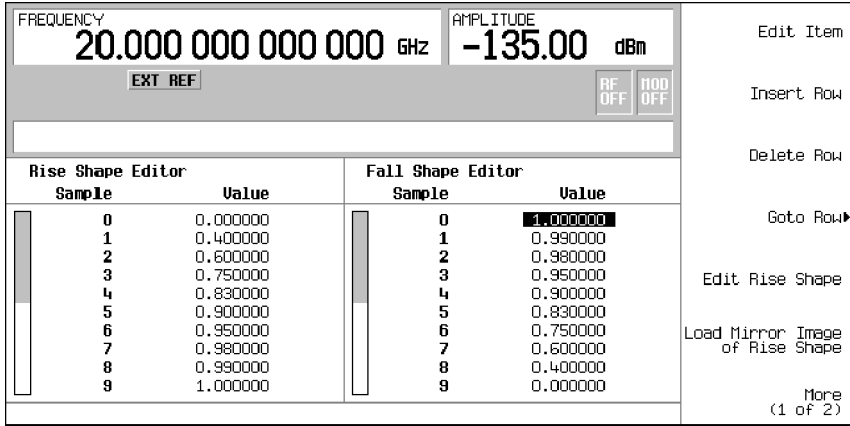
5. Press **More (1 of 2)** > **Display Burst Shape**.

This displays a graphical representation of the waveform's rise and fall characteristics.

Figure 6-12
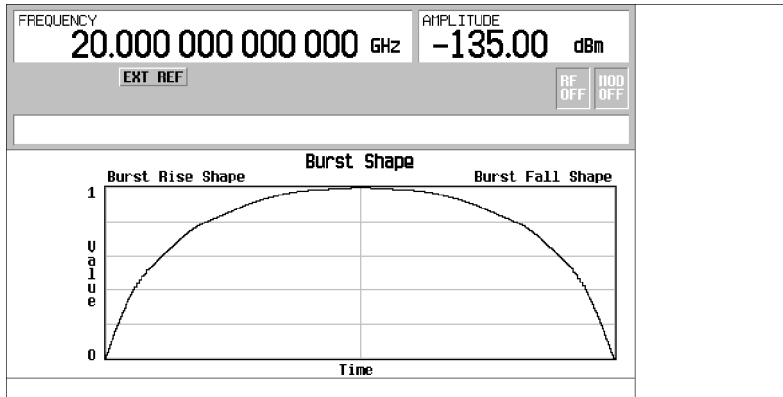
NOTE	To return the burst shape to the default conditions, press **Return** > **Return** > **Confirm Exit From Table Without Saving** > **Restore Default Burst Shape**.

6. Press **Return** > **Load/Store** > **Store To File**.

   If there is already a file name from the Catalog of SHAPE Files occupying the active entry area, press the following keys:

   Editing Keys > **Clear Text**

7. Enter a file name (for example, NEWBURST) using the alpha keys and the numeric keypad.

8. Press **Enter**.

   The contents of the current Rise Shape and Fall Shape editors are stored to the Catalog of SHAPE Files. This burst shape can now be used to customize a modulation or as a basis for a new burst shape design.

To Select and Recall a User-Defined Burst Shape Curve from the Memory Catalog

Once a user-defined burst shape file is stored in the Memory Catalog, it can be recalled for use with real-time I/Q baseband generated digital modulation.

This example requires a user-defined burst shape file stored in memory. If you have not created and stored a user-defined burst shape file, complete the steps in the previous sections.

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Burst Shape** > **Burst Shape Type** > **User File**.

3. Highlight the desired burst shape file (for example, NEWBURST).

4. Press **Select File**.

   The selected burst shape file is now applied to the current real-time I/Q baseband digital modulation state.

5. Press **Return** > **Custom Off On**.

   This generates the custom modulation with user-defined burst shape created in the previous steps. During waveform generation, the CUSTOM and I/Q annunciators activate. The waveform is now modulating the RF carrier.

6. Press **RF On/Off**.

   The current real-time I/Q baseband digital modulation format with user-defined burst shape should be available at the signal generator's RF OUTPUT connector.

Working with Configuration of Hardware

In this section, you will learn about the following:

To Set the BBG Reference to External or Internal

1. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Configure Hardware**.

   Configure Hardware allows you to access a menu from which you can set the BBG Reference is set to External or Internal.

2. Press **BBG Ref Ext Int** to select either external or internal as the bit-clock reference for the data generator.

   If the external choice is selected, the external frequency value must be applied to the BASEBAND GEN REF IN rear panel connector.

To Set the BBG Reference External Frequency

The BBG reference external frequency is only used when the **BBG Ref Ext Int** softkey has been set to Ext (external).

1. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Configure Hardware**.

   Configure Hardware allows you to access a menu from which you can set the external BBG reference frequency.

2. Press **Ext BBG Ref Freq**.

3. Using the numeric keypad, enter a desired frequency and press **MHz**, **kHz**, or **Hz**.

To Set the External DATA CLOCK to Receive Input as Either Normal or Symbol

1. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Configure Hardware**.

   Configure Hardware allows you to access a menu from which you can set the external DATA CLOCK to receive input as either Normal or Symbol.

2. Press **Ext Data Clock** to select either Normal or Symbol; this setting has no effect in internal clock mode.

   • When set to Normal, the DATA CLOCK input connector requires a bit clock.

   • When set to Symbol, a one-shot or continuous symbol sync signal must be provided to the SYMBOL SYNC input connector.

To Set the BBG DATA CLOCK to External or Internal

1. Press Mode > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Configure Hardware**.

   Configure Hardware allows you to access a menu from which you can set the BBG DATA CLOCK to receive input from External or Internal.

2. Press **BBG Data Clock Ext Int** to select either external or internal.

   • When set to Ext (external), the DATA CLOCK connector is used to supply the BBG Data Clock.

   • When set to Int (internal), the internal data clock is used.

To Adjust the I/Q Scaling

Adjusting the I/Q Scaling (amplitude of the I/Q outputs) multiplies the I and Q data by the I/Q scaling factor that is selected and can be used to improve the Adjacent Channel Power (ACP). Lower scaling values equate to better ACP. This setting has no effect with MSK or FSK modulation.

1. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Configure Hardware**.

   Configure Hardware allows you to access a menu from which you can adjust the I/Q Scaling.

2. Press **I/Q Scaling**, enter a desired I/Q scaling level, and press **%**.

Working with Phase Polarity

To Set Phase Polarity to Normal or Inverted

1. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **More (1 of 3)** > **Phase Polarity Normal Invert**.

   Phase Polarity Normal Invert allows you to leave the selection as Normal (so that the phase relationship between the I and Q signals is not altered by the phase polarity function) or set to Invert and invert the internal Q signal, reversing the rotation direction of the phase modulation vector.

   When you choose Invert, the in-phase component lags the quadrature-phase component by 90° in the resulting modulation. Inverted phase polarity is required by some radio standards and it is useful for lower sideband mixing applications. The inverted selection also applies to the I, I-bar, Q, and Q-bar output signals.

Working with Differential Data Encoding

Diff Data Encode Off On allows you to toggle the operational state of the signal generator's differential data encoding.

- When set to Off, data bits are not encoded prior to modulation.

- When set to On, data bits are encoded prior to modulation. Differential encoding uses an exclusive-OR function to generate a modulated bit. Modulated bits will have a value of 1 if a data bit is different from the previous bit or they will have a value of 0 if a data bit is the same as the previous bit.

In this section, you will learn about the following:

- "Understanding Differential Encoding" on page 171
- "To Use Differential Encoding" on page 177

Understanding Differential Encoding

Differential encoding is a digital-encoding technique whereby a binary value is denoted by a signal *change* rather than a particular signal state. Using differential encoding, binary data in any user-defined I/Q or FSK modulation can be encoded during the modulation process via symbol table offsets defined in the Differential State Map.

For example, consider the signal generator's default 4QAM I/Q modulation. With a user-defined modulation based on the default 4QAM template, the `I/Q Values` editor contains data that represent four symbols (00, 01, 10, and 11) mapped into the I/Q plane using two distinct values, 1.000000 and -1.000000. These four symbols can be differentially encoded during the modulation process by assigning symbol table offset values associated with each data value. Figure 6-13 shows the 4QAM modulation in the `I/Q Values` editor.

Figure 6-13
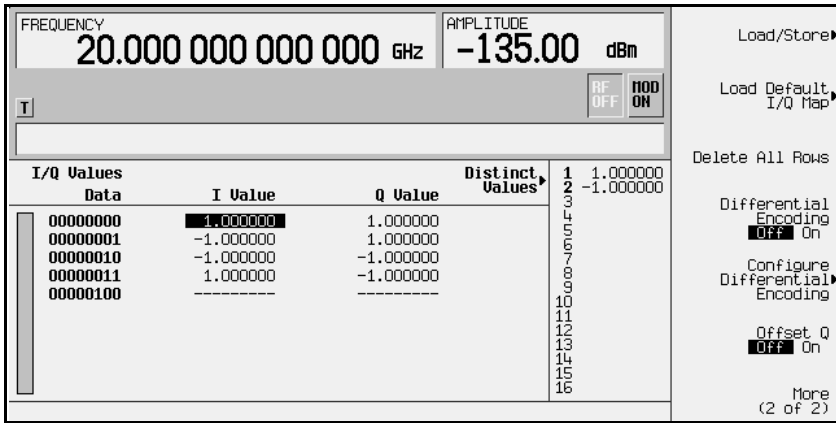
NOTE	The number of bits per symbol can be expressed using the following formula. Because the equation is a ceiling function, if the value of x contains a fraction, x is rounded up to the next whole number.

$$x = \lceil Log_2(y)$$

Where x = bits per symbol, and y = the number of differential states.

The following illustration shows a 4QAM modulation I/Q State Map.

2nd Symbol
Data = 00000001
Distinct values: -1, +1

1st Symbol
Data = 00000000
Distinct values: +1, +1

I/Q State Map
I/Q States: 4

3rd Symbol
Data = 00000010
Distinct values: -1, -1

4th Symbol
Data = 00000011
Distinct values: +1, -1
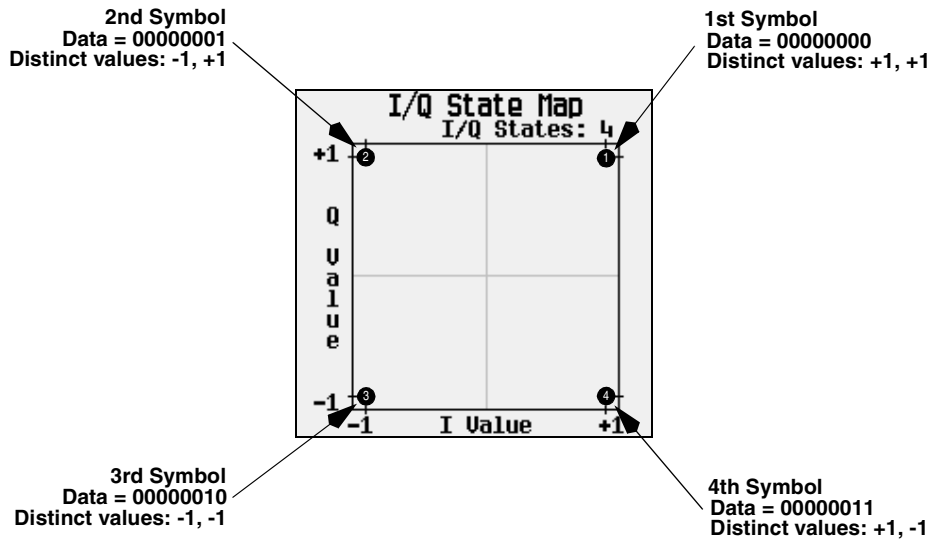
Differential Data Encoding

In real-time I/Q baseband digital modulation waveforms, data (1's and 0's) are encoded, modulated onto a carrier frequency and subsequently transmitted to a receiver. In contrast to differential encoding, differential *data* encoding modifies the data stream *prior* to I/Q mapping. Where differential encoding encodes the raw data by using symbol table offset values to manipulate I/Q mapping at the point of modulation, differential data encoding uses *the transition from one bit value to another* to encode the raw data.

Differential data encoding modifies the raw digitized data by creating a secondary, encoded data stream that is defined by *changes* in the digital state, from 1 to 0 or from 0 to 1, of the raw data stream. This differentially encoded data stream is then modulated and transmitted.

In differential data encoding, a *change* in a raw data bit's digital state, from 1 to 0 or from 0 to 1, produces a 1 in the encoded data stream. *No change* in digital state from one bit to the next, in other words a bit with a value of 1 followed by another bit with a value of 1 or a bit with a value of 0 followed by the same, produces a 0 in the encoded data. For instance, differentially encoding the data stream containing 01010011001010 renders 1111010101111.

Differential data encoding can be described by the following equation:

$$transmittedbit(i) = databit(i-1) \oplus databit(i)$$

For a bit-by-bit illustration of the encoding process, see the following illustration:
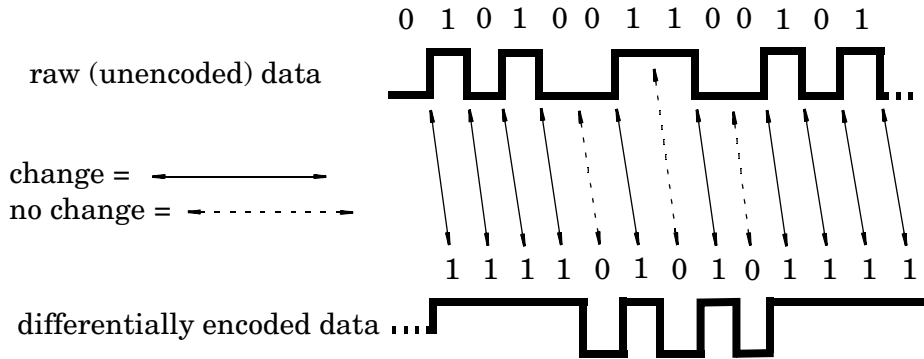
How Differential Encoding Works

Differential encoding employs offsets in the symbol table to encode user-defined modulation schemes. The `Differential State Map` editor is used to introduce symbol table offset values which in turn cause transitions through the I/Q State Map based on their associated data value. Whenever a data value is modulated, the offset value stored in the Differential State Map is used to encode the data by transitioning through the I/Q State Map in a direction and distance defined by the symbol table offset value.

Entering a value of +1 will cause a 1-state forward transition through the I/Q State Map, as shown in the following illustration.

NOTE	The following I/Q State Map illustrations show all of the possible state transitions using a particular symbol table offset value. The actual state-to-state transition would depend upon the state in which the modulation had started.

As an example, consider the following data/symbol table offset values.

Table 6-2

Data	Offset Value
00000000	+1
00000001	-1
00000010	+2
00000011	0

These symbol table offsets will result in one of the transitions, as shown.

**Data Value 00000000
with Symbol Table Offset +1
transition 1 state forward**

**Data Value 00000001
with Symbol Table Offset -1
transition 1 state backward**

**Data Value 00000010
with Symbol Table Offset +2
transition 2 states forward**

**Data Value 00000011
with Symbol Table Offset 0
no transition**
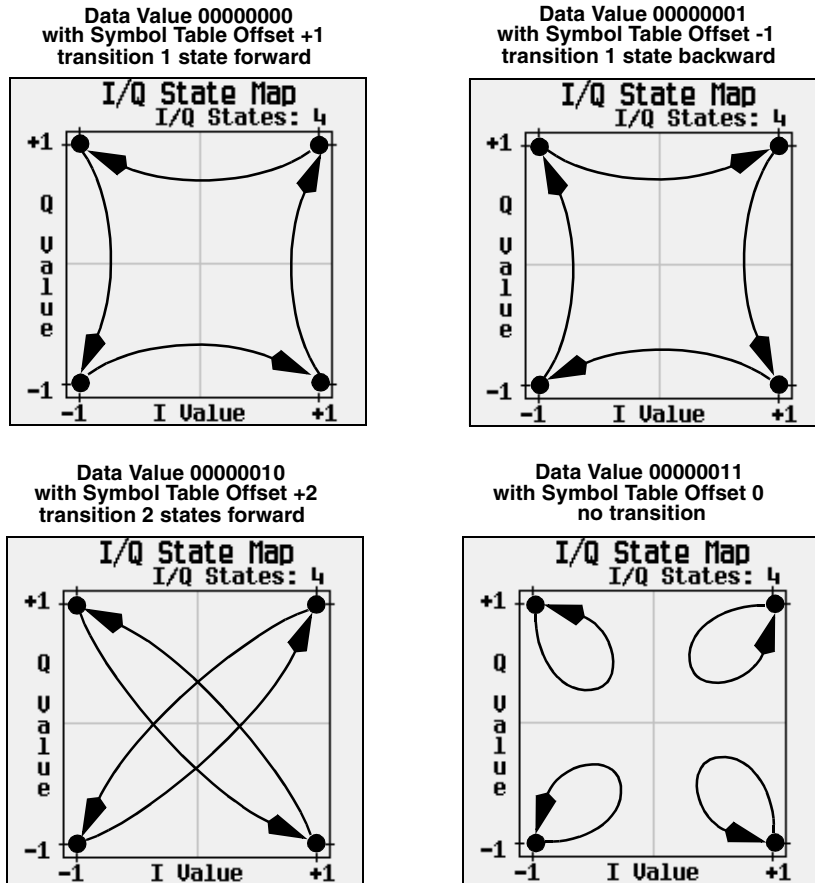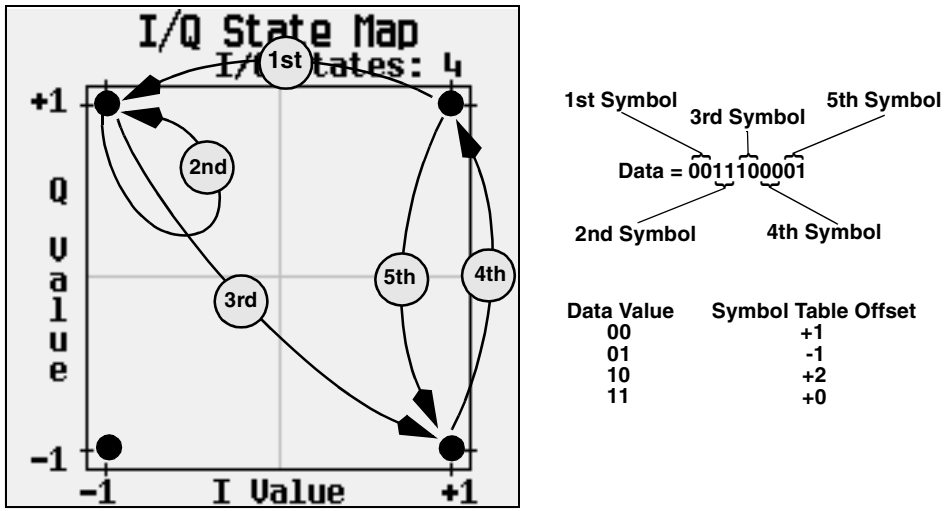
When applied to the user-defined default 4QAM I/Q map, starting from the 1st symbol (data 00), the differential encoding transitions for the data stream (in 2-bit symbols) 0011100001 appear in the previous illustration.

As you can see, the 1st and 4th symbols, having the same data value (00), produce the same state transition (forward 1 state). In differential encoding, symbol values do not define location; they define the direction and distance of a *transition* through the I/Q State Map.

For instructions on configuring differential encoding, see "Understanding Differential Encoding" on page 171.

To Use Differential Encoding

Differential encoding is a digital-encoding technique that denotes a binary value by a signal *change* rather than a particular signal state. It is available for Custom Real Time I/Q Baseband mode. It is not available for waveforms generated by Arb Waveform Generator mode.

The signal generator's `Differential State Map` editor enables you to modify the differential state map associated with user-defined I/Q and user-defined FSK modulations. In this procedure, you create a user-defined I/Q modulation and then configure, activate, and apply differential encoding to the user-defined modulation. For more information, see "Understanding Differential Encoding" on page 171.

This section teaches you how to perform the following tasks:

- "Configuring User-Defined I/Q Modulation" on page 177
- "Accessing the Differential State Map Editor" on page 178
- "Editing the Differential State Map" on page 179
- "Activating Differential Data Encoding" on page 180

Configuring User-Defined I/Q Modulation

1. Press **Preset**.

2. Press **Mode** > **Custom** > **Real Time I/Q Baseband** > **Modulation Type** > **Define User I/Q** > **More (1 of 2)** > **Load Default I/Q Map** > **QAM** > **4QAM**.

This loads a default 4QAM I/Q modulation and displays it in the I/Q Values editor.

The default 4QAM I/Q modulation contains data that represent 4 symbols (00, 01, 10, and 11) mapped into the I/Q plane using 2 distinct values (1.000000 and −1.000000). These 4 symbols will be traversed during the modulation process by the symbol table offset values associated with each symbol of data.

Figure 6-14
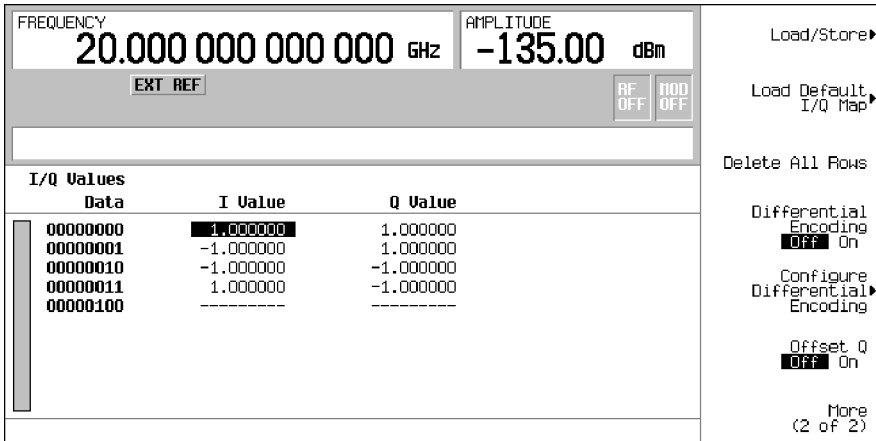
Accessing the Differential State Map Editor

1. Press **Configure Differential Encoding**.

This opens the Differential State Map editor. At this point, you see the data for the 1st symbol (00000000) and the cursor prepared to accept an offset value. You are now prepared to create a custom differential encoding for the user-defined default 4QAM I/Q modulation.

Figure 6-15
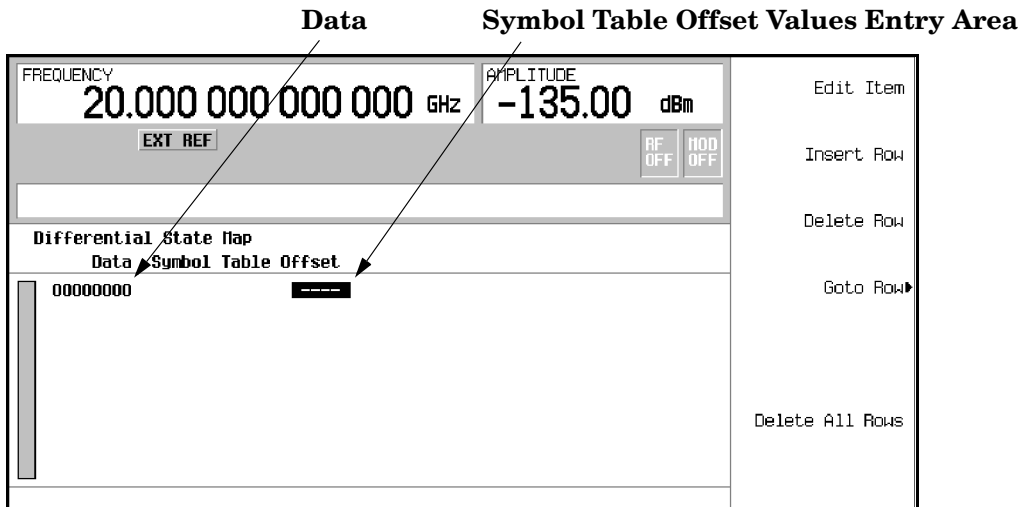
Editing the Differential State Map

1. Press **1** > **Enter**.

   This encodes the first symbol by adding a symbol table offset of 1. The symbol rotates *forward* through the state map by 1 value when a data value of 0 is modulated.

2. Press **+/-** > **1** > **Enter**.

   This encodes the second symbol by adding a symbol table offset of -1. The symbol rotates *backward* through the state map by 1 value when a data value of 1 is modulated.

NOTE	At this point, the modulation has one bit per symbol. For the first two data values (00000000 and 00000001) only the last bits (the 0 and the 1, respectively) are significant.

---

3. Press **2** > **Enter**.

   This encodes the third symbol by adding a symbol table offset of 2. The symbol rotates *forward* through the state map by 2 values when a data value of 10 is modulated.

4. Press **0** > **Enter**.

   This encodes the fourth symbol by adding a symbol table offset of 0. The symbol does *not* rotate through the state map when a data value of 11 is modulated.

NOTE	At this point, the modulation has two bits per symbol. For the data values 00000000, 00000001, 00000010, 00000011, the symbol values are 00, 01, 10, and 11 respectively.

---

5. Press **Return** > **Differential Encoding Off On**.

   This applies the custom differential encoding to a user-defined modulation.

NOTE	Notice that (UNSTORED) appears next to Differential State Map on the signal generator's display. Differential state maps are associated with the user-defined modulation for which they were created.
	In order to save a custom differential state map, you must store the user-defined modulation for which it was designed. Otherwise the symbol table offset data is purged when you press the **Confirm Exit From Table Without Saving** softkey when exiting from the I/Q or FSK editor.

Activating Differential Data Encoding

1. Press **Return**.

2. Press **More (1 of 3)** > **Diff Data Encode Off On**.

This activates differential data encoding for the current real-time I/Q baseband digital modulation format.

To generate and output the custom digital modulation, complete the steps in the following sections.

7   Dual Arbitrary Waveform Generator

This chapter describes the Dual Arb mode, which is available only in E8267C PSG vector signal generators with Option 002/602.

This chapter includes the following major sections:

Using the Dual ARB Waveform Player

The dual arbitrary (ARB) waveform player is used for editing and playing waveform files. Waveform files consist of two types: segments (WFM1) and sequences (SEQ). Segments are individual waveforms that are defined using an installed ARB format, such as Two Tone, and created using the internal arbitrary waveform generator. Sequences are several individual segments strung together in one file. Waveform files can also be created remotely and downloaded to the PSG for playback as a segment. For information on downloading waveforms, see "Programming and Downloading Waveforms" on page 207.

A waveform is generated when an ARB modulation format is turned on. The resultant segment file is automatically named AUTOGEN_WAVEFORM. Since this default file name is shared with other ARB formats, the file should be renamed in the dual ARB player after turning the modulation format off. If you do not rename the file, it will be overwritten when the same or another ARB format is turned on.

Waveform player features include waveform clipping, markers, and triggering. Clipping allows you to reduce high power peaks, which can cause adjacent channel noise. Markers and triggering are useful for synchronizing the output of the signal generator with other devices.

Before you can work with a waveform file, it must reside in volatile memory. A newly generated segment file (AUTOGEN_WAVEFORM) initially resides in volatile memory until you store it to non-volatile memory. Whenever you cycle the power on the PSG or download new firmware, you must reload your waveform file from non-volatile memory.

To Create and Play Waveform Segments

This procedure shows you how to create and play back waveform segments using internally generated two tone and multitone waveforms. In a later procedure, you will use both segments to build a waveform sequence. This procedure comprises the following tasks:

- "Generating a Waveform" on page 183
- "Creating a Waveform Segment" on page 183
- "Playing the Waveform Segment" on page 184
- "Generating the Second Waveform" on page 184
- "Creating the Second Waveform Segment" on page 184

Generating a Waveform

1. Press **Preset**.

2. Press **Mode > Two Tone**.

3. Press **Alignment Left Cent Right** to Right.

4. Press **Two Tone Off On** to On.

5. Press **Two Tone Off On** to Off.

   This generates a two tone waveform with the tone on the right placed at the carrier frequency. During waveform generation, the T-TONE and I/Q annunciators activate. The waveform is stored in volatile memory with the default file name AUTOGEN_WAVEFORM, as you will see in the next section. The Two Tone mode was turned off after generation because a waveform cannot be renamed as a segment while it is in use.

NOTE	There can only be one AUTOGEN_WAVEFORM waveform in memory at any given time. Therefore, you must rename this file, clearing the way for a second waveform.

Creating a Waveform Segment

1. Press **Mode > Dual ARB**.

2. Press **Waveform Segments**.

3. Press **Load Store** to Store.

4. Highlight the default segment AUTOGEN_WAVEFORM.

5. Press **More (1 of 2) > Rename Segment > Editing Keys > Clear Text**.

6. Enter a file name (for example, TTONE) using the alpha keys and the numeric keypad.

7. Press **Enter**.

The waveform segment is renamed and remains in volatile memory as a WFM1 file.

Playing the Waveform Segment

1. Press **Mode** > **Dual ARB** > **Select Waveform**. (If you are continuing from the previous task, you can also press **Mode Setup** to return to the top-level Dual ARB menu.)

2. Highlight the waveform segment you just created (for example, TTONE).

3. Press **Select Waveform**.

4. Press **ARB Off On** to On.

The selected waveform is played and the ARB and I/Q annunciators activate. Only waveform files residing in volatile memory can be selected for playback. If a file has been stored to non-volatile memory, it must first be loaded. Refer to "Loading Waveform Segments from Non-volatile Memory" on page 185.

Generating the Second Waveform

1. Press **Mode** > **Multitone**.

2. Press **Initialize Table** > **Number Of Tones** > **9** > **Enter** > **Done**.

3. Press **Multitone Off On** to On.

4. Press **Multitone Off On** to Off.

This generates a new multitone waveform with nine tones. During waveform generation, the M−TONE and I/Q annunciators activate. The waveform is stored in volatile memory with the default file name AUTOGEN_WAVEFORM. The Multitone mode was turned off after generation because a waveform cannot be renamed as a segment while it is in use.

Creating the Second Waveform Segment

1. Press **Mode** > **Dual ARB**.

2. Press **Waveform Segments**.

3. Press **Load Store** to Store.

4. Highlight the default segment AUTOGEN_WAVEFORM.

5. Press **More (1 of 2)** > **Rename Segment** > **Editing Keys** > **Clear Text**.

6. Enter a file name (for example, MTONE) using the alpha keys and the numeric keypad.

7. Press **Enter**.

The second waveform segment is renamed and remains in volatile memory as a WFM1 file. The two segments created in this procedure will be used to build a waveform sequence in a later procedure.

To Store and Load Waveform Segments

Waveform segments can reside in volatile memory as WFM1 files, or they can be stored to non-volatile memory as NVWFM files, or both. To play or edit a waveform file, it must reside in volatile memory. Files stored only in volatile memory, however, will not survive a power cycle. Therefore, it is a good practice to store important files to non-volatile memory and load them to volatile memory whenever you want to use them.

Storing Waveform Segments to Non-volatile Memory

1. Press **Mode > Dual ARB**.

2. Press **Waveform Segments**.

3. Press **Load Store** to Store, if necessary.

4. Press **Store All To NVWFM Memory**.

Copies of all WFM1 waveform segment files have been stored in non-volatile memory as NVWFM files. You can also store files individually by highlighting the file and pressing
Store Segment To NVWFM Memory.

Loading Waveform Segments from Non-volatile Memory

1. Power cycle the instrument.

   This will clear out the volatile memory and delete all WFM1 files.

2. Press **Mode > Dual ARB**.

3. Press **Waveform Segments**.

4. Press **Load Store** to Load, if necessary.

5. Press **Load All From NVWFM Memory**.

Copies of all NVWFM waveform segment files have been loaded into volatile memory as WFM1 files. You can also load files individually by highlighting the file and pressing
Load Segment From NVWFM Memory.

To Build and Edit a Waveform Sequence

In this example, you learn how to build and edit a waveform sequence using two waveform segments. If you have not created the waveform segments used to build a waveform sequence, complete the steps in the previous section, "To Create and Play Waveform Segments" on page 182.

Creating a Waveform Sequence Using Waveform Segments

1. Press **Mode** > **Dual ARB** > **Waveform Sequences**
2. Press **Build New Waveform Sequence** > **Insert Waveform**.
3. Highlight the first waveform segment (for example, TTONE).
4. Press **Insert Selected Waveform**.
5. Highlight the second waveform segment (for example, MTONE).
6. Press **Insert Selected Waveform**.
7. Press **Done Inserting**.
8. Press **Name and Store**.
9. Enter a file name (for example, TTONE+MTONE) using the alpha keys and the numeric keypad.
10. Press **Enter**.

You have now defined the sequence as one repetition of the two-tone waveform segment followed by one repetition of the nine-tone multitone waveform segment. The sequence has been stored under a new name to the Catalog of Seq Files in the signal generator's memory catalog.

To play the waveform, refer to "Playing the Waveform Segment" on page 184. The same steps apply.

Editing Waveform Segment Repetition

1. Press **Waveform Sequences > Edit Selected Waveform Sequence**.

2. Highlight the first waveform segment entry (for example, WFM1:TTONE).

3. Press **Edit Repetitions > 100 > Enter**.

4. Press **Edit Repetitions > 200 > Enter**.

5. Press **Name And Store**.

6. Enter a file name (for example, TTONE100+MTONE200) using the alpha keys and the numeric keypad.

7. Press **Enter**.

You have now changed the number of repetitions for each waveform segment entry from 1 to 100 and 200, respectively. The sequence has been stored under a new name to the Catalog of Seq Files in the signal generator's memory catalog.

To play the waveform sequence, refer to . The same steps apply.

Using Waveform Clipping

Clipping limits power peaks in waveform segments by clipping the I and Q data to a selected percentage of its highest peak. *Circular* clipping is defined as clipping the composite I/Q data (I and Q data are equally clipped). *Rectangular* clipping is defined as independently clipping the I and Q data. For more information, see "Waveform Clipping Concepts" on page 189.

In this section, you learn how to clip waveform segments. If you have not created waveform segments, complete the steps in the previous section, "To Create and Play Waveform Segments" on page 182.

To Configure Circular Clipping

1. Press **Mode** > **Dual ARB** > **Waveform Segments**.

2. Press **Load Store** to Store.

3. Highlight the first waveform segment (for example, TTONE).

4. Press **Waveform Utilities** > **Clipping**.

5. Press **Clip |I+jQ| To** > **80** > **%** > **Apply to Waveform**.

    The I and Q data are both clipped by 80%. You will see 80.0% displayed below the **Clip |I+jQ| To** softkey.

To Configure Rectangular Clipping

1. Press **Mode** > **Dual ARB** > **Waveform Segments**.

2. Press **Load Store** to Store.

3. Highlight the second waveform segment (for example, MTONE).

4. Press **Waveform Utilities** > **Clipping**.

5. Press **Clipping Type |I+jQ| |I|,|Q|**.

    This activates the **Clip |I| To** and **Clip |Q| To** softkeys that allow you to configure rectangular (independent) I and Q data clipping.

6. Press **Clip |I| To** > **80** > **%**.

7. Press **Clip |Q| To** > **40** > **%** > **Apply to Waveform**.

    The I and Q data are individually clipped by 80% and 40%, respectively. You will see 80.0% displayed below the **Clip |I| To** softkey and 40.0% below the **Clip |Q| To** softkey.

Waveform Clipping Concepts

Waveforms with high power peaks can cause intermodulation distortion, which generates spectral regrowth (a condition that interferes with signals in adjacent frequency bands). The clipping function allows you to reduce high power peaks.

The clipping feature is available only with the Dual Arb mode.

How Power Peaks Develop

To understand how clipping reduces high power peaks, it is important to know how the peaks develop as the signal is constructed.

I/Q waveforms can be the summation of multiple channels (refer to Figure 7-1). Whenever most or all of the individual channel waveforms simultaneously contain a bit in the same state (high or low), an unusually high power peak (negative or positive) occurs in the summed waveform. This does *not* happen frequently because the high and low states of the bits on these channel waveforms are random, which causes a cancelling effect.

Figure 7-1   **Multiple Channel Summing**
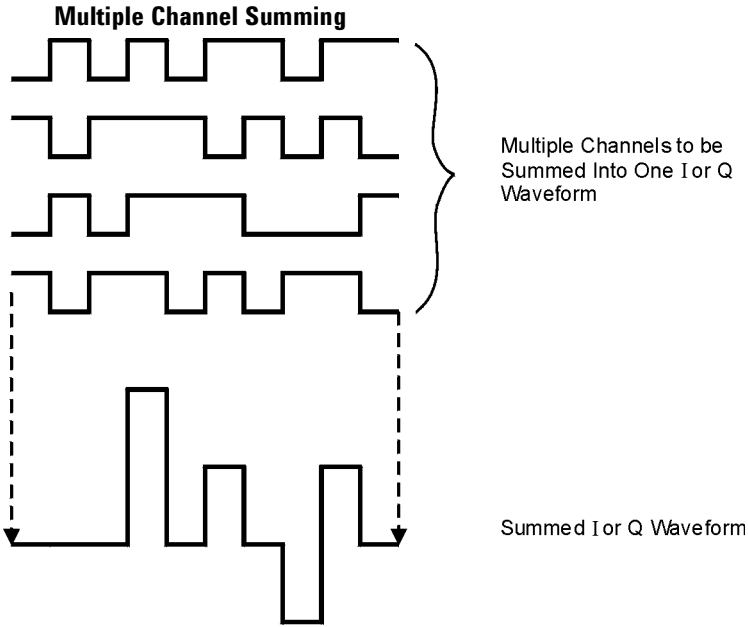
Multiple Channels to be
Summed Into One I or Q
Waveform

Summed I or Q Waveform

pk722b

The I and Q waveforms combine in the I/Q modulator to create an RF waveform. The magnitude of the RF envelope is determined by the equation $\sqrt{I^2+Q^2}$, where the squaring of I and Q always results in a positive value. Notice how simultaneous positive and negative peaks in the I and Q waveforms do not cancel each other, but combine to create an even greater peak (refer to Figure 7-2).

Figure 7-2   **Combining the I and Q Waveforms**
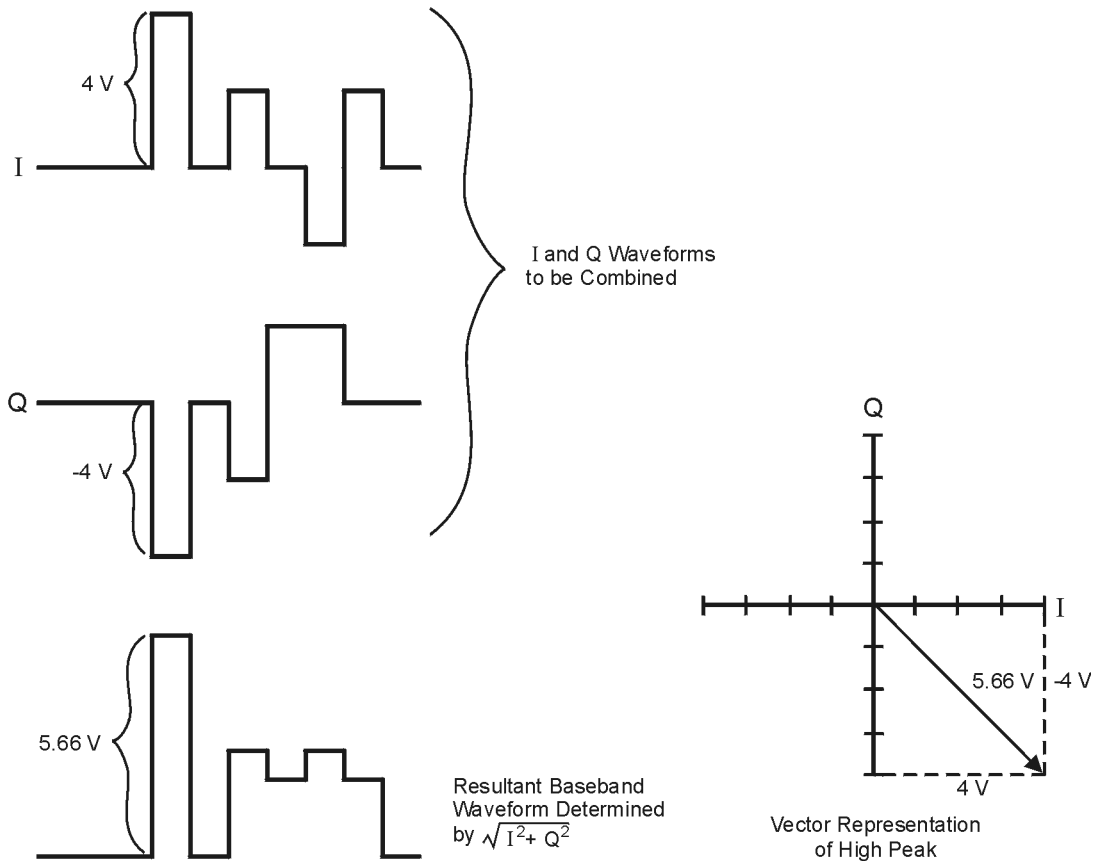
Pk750b

How Peaks Cause Spectral Regrowth

Because of the relative infrequency of high power peaks, a waveform will have a high peak-to-average power ratio (refer to Figure 7-3). Because a transmitter's power amplifier gain is set to provide a specific average power, high peaks can cause the power amplifier to move toward saturation. This causes intermodulation distortion, which generates spectral regrowth.

Figure 7-3   **Peak-to-Average Power**
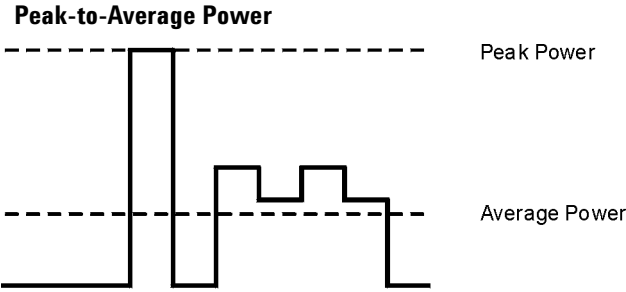
Spectral regrowth is a range of frequencies that develops on each side of the carrier (similar to sidebands) and extends into the adjacent frequency bands (refer to Figure 7-4). Consequently, spectral regrowth interferes with communication in the adjacent bands. Clipping can provide a solution to this problem.

Figure 7-4   **Spectral Regrowth Interfering with Adjacent Band**
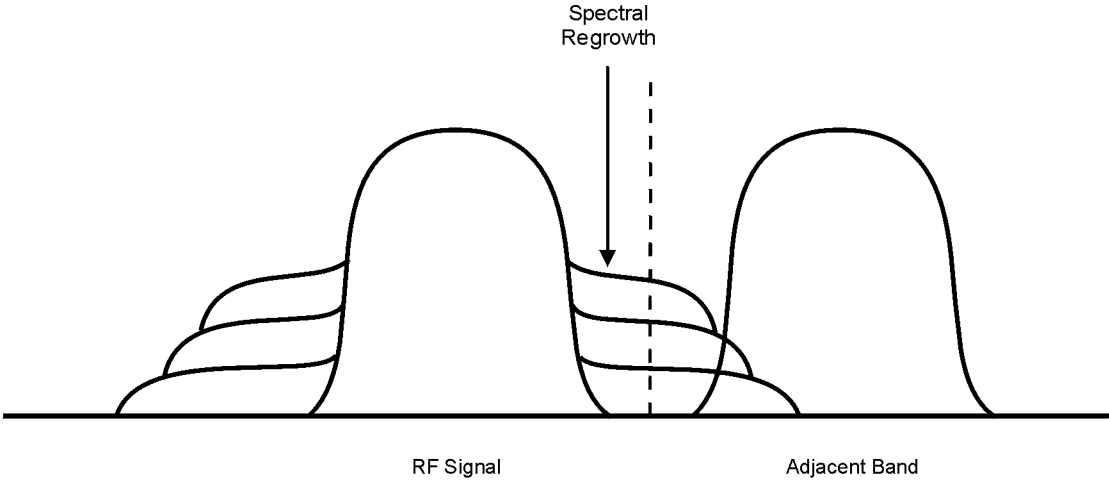
How Clipping Reduces Peak-to-Average Power

You can reduce peak-to-average power, and consequently spectral regrowth, by clipping the waveform to a selected percentage of its peak power. The PSG vector signal generator provides two different methods of clipping: circular and rectangular.

During *circular* clipping, clipping is applied to the combined I and Q waveform (|I + jQ|). Notice in Figure 7-5 that the clipping level is constant for all phases of the vector representation and appears as a circle. During *rectangular* clipping, clipping is applied to the I and Q waveforms separately (|I|, |Q|). Notice in Figure 7-6 on page 193 that the clipping level is different for I and Q; therefore, it appears as a rectangle in the vector representation. With either method, the objective is to clip the waveform to a level that effectively reduces spectral regrowth, but does *not* compromise the integrity of the signal. Figure 7-7 on page 194 uses two complementary cumulative distribution plots to show the reduction in peak-to-average power that occurs after applying circular clipping to a waveform.

The lower you set the clipping value, the lower the peak power that is passed (or the more the signal is clipped). Often, the peaks can be clipped successfully without substantially interfering with the rest of the waveform. Data that might be lost in the clipping process is salvaged because of the error correction inherent in the coded systems. If you clip too much of the waveform, however, lost data is irrecoverable. You may have to try several clipping settings to find a percentage that works well.

Figure 7-5   **Circular Clipping**
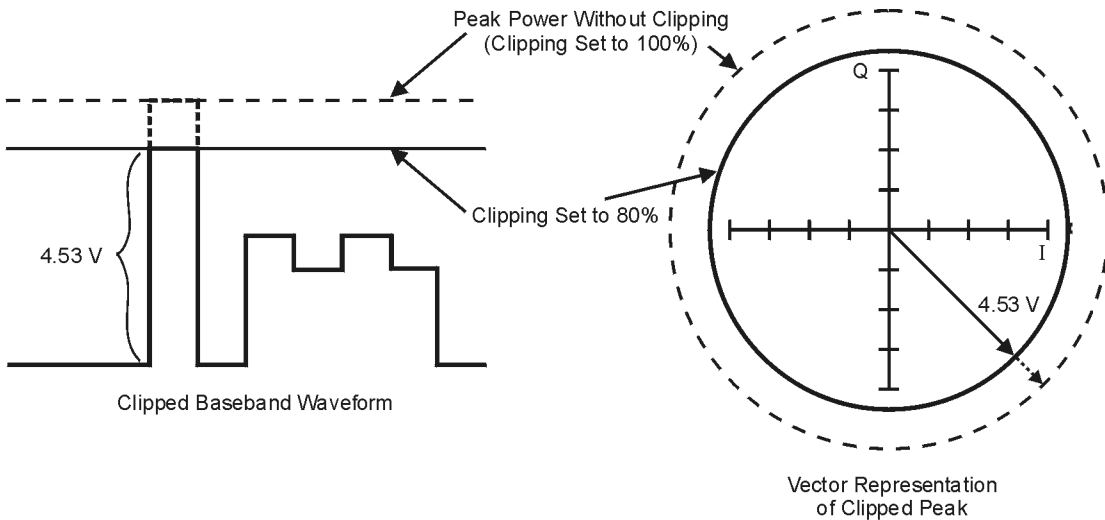
Pk748b

Figure 7-6   Rectangular Clipping

Clipped I Waveform

Clipped Q Waveform

Vector Representation
of Clipped I & Q Peak
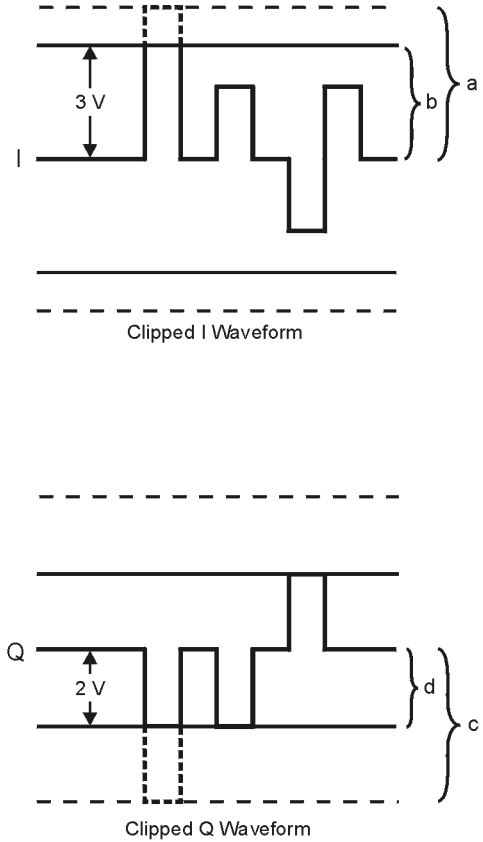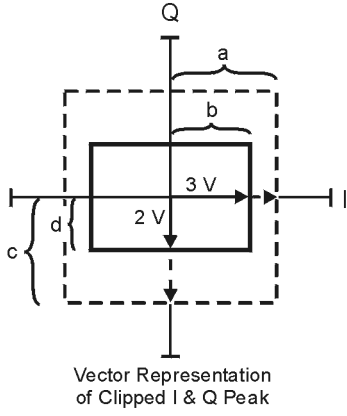
a) |I| Clipping Set to 100% (No Clipping)

b) |I| Clipping Set to 75% of Greatest Peak

c) |Q| Clipping Set to 100% (No Clipping)

d) |Q| Clipping Set to 50% of Greatest Peak

pk751b

Figure 7-7   Reduction of Peak-to-Average Power

Complementary Cumulative Distribution

CCDF with Clipping = 100% (No Clipping)

CCDF with Clipping = 80%
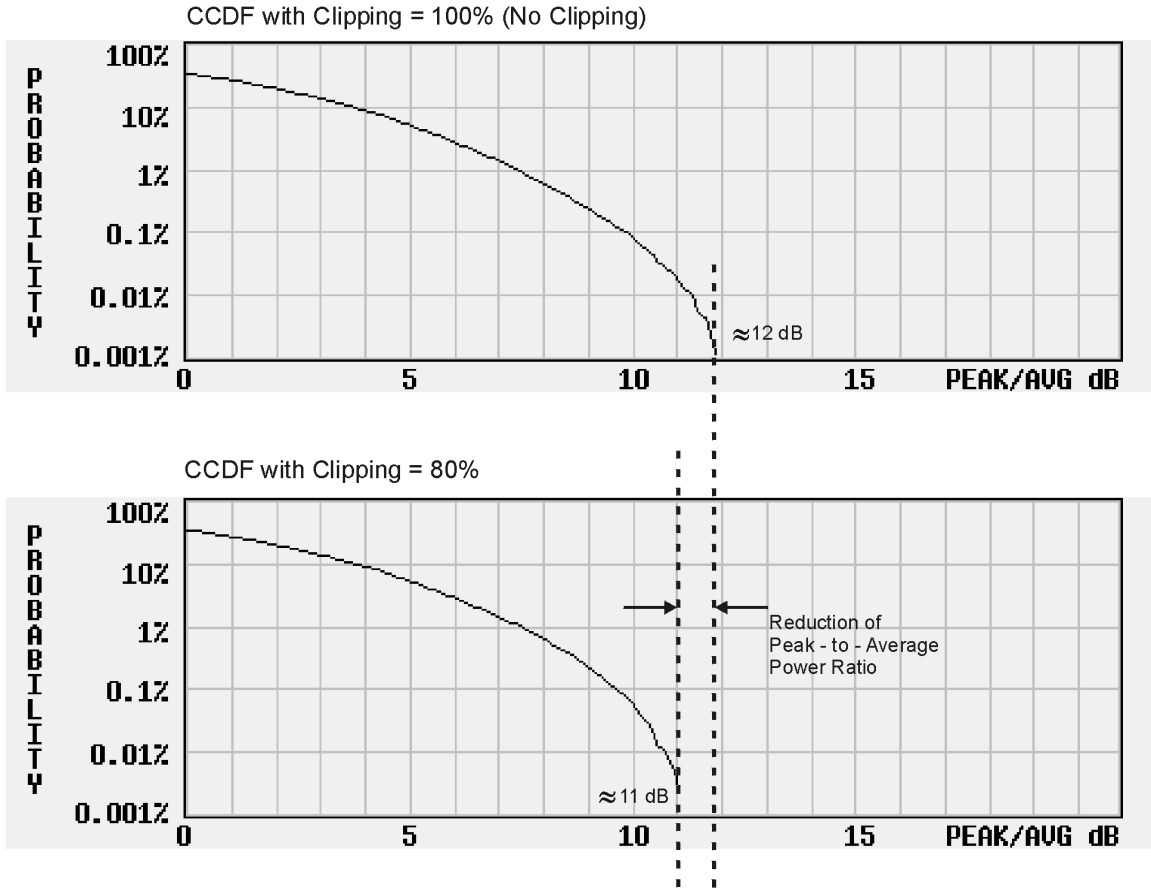
Pk734b

Using Waveform Markers

Waveform markers provide auxiliary output signals that are synchronized with a waveform segment. You can place up to four markers on a waveform segment. However, only Marker 1 and Marker 2 can be placed using the waveform player's user interface (for more information, refer to "Waveform Marker Concepts" on page 201).

Using markers, you can construct an output signal as a trigger to synchronize another instrument to a given portion of a waveform. You can also place markers into a waveform sequence, either as the sequence is being built or within an existing waveform sequence.

To Place a Marker at the First Point within a Waveform Segment

If you have not created a waveform segment, complete the steps in the previous sections, "Generating a Waveform" on page 183 and "Creating a Waveform Segment" on page 183.

1. Press **Mode** > **Dual ARB** > **Waveform Segments**.

2. Press **Load Store**.

3. Highlight a waveform segment (for example, TTONE).

4. Press **Waveform Utilities** > **Set Markers** > **Set Marker On First Point**.

This sets Marker 1 (selected by default) on the first point in the selected waveform segment. For instructions on verifying marker operation, see "To Verify Marker Operation" on page 200.

To Place a Marker Across a Range of Points within a Waveform Segment

If you have not created a waveform segment, complete the steps in the previous sections, "Generating a Waveform" on page 183 and "Creating a Waveform Segment" on page 183.

1. Press **Mode** > **Dual ARB** > **Waveform Segments**.

2. Press **Load Store**.

3. Highlight a waveform segment (for example, TTONE).

4. Press **Waveform Utilities** > **Set Markers** > **Set Marker On Range Of Points**.

5. Press **First Mkr Point** > **10** > **Enter**.

6. Press **Last Mkr Point** > **163830** > **Enter**.

7. Press **Apply To Waveform**.

NOTE	The last marker point must be greater than or equal to the first marker point.

This activates Marker 1 (selected by default) from point 10 to point 163830 in the selected waveform segment.

For instructions on verifying marker operation, see "To Verify Marker Operation" on page 200.

To Place Repetitively Spaced Markers within a Waveform Segment

If you have not created a waveform segment, complete the steps in the previous sections, "Generating a Waveform" on page 183 and "Creating a Waveform Segment" on page 183.

1. Press **Mode > Dual ARB > Waveform Segments**.

2. Press **Load Store**.

3. Highlight a waveform segment (for example, TTONE).

4. Press **Waveform Utilities > Set Markers > Set Marker On Range Of Points**.

5. Press **First Mkr Point > 10 > Enter**.

6. Press **Last Mkr Point > 163830 > Enter**.

7. Press **# Skipped Points > 2 > Enter**.

8. Press **Apply To Waveform**.

NOTE	The last marker point must be greater than or equal to the first marker point.

This activates Marker 1 (selected by default) every three points from point 10 to point 163830 in the selected waveform segment.

For instructions on verifying marker operation, see "To Verify Marker Operation" on page 200.

To Use Marker 2 to Blank the RF Output

If you have not created a waveform segment, complete the steps in the previous sections, "Generating a Waveform" on page 183 and "Creating a Waveform Segment" on page 183.

NOTE	RF blanking applies to Marker 2 only. Marker 1 does not blank the RF output. For more information, see "Waveform Marker Concepts" on page 201.

1. Press **Preset**.

2. Press **Mode** > **Dual ARB** > **Select Waveform**.

3. Highlight a waveform segment (for example, TTONE).

4. Press **Select Waveform**.

5. Press **Mode** > **Dual ARB** > **ARB Setup** > **Mkr 2 To RF Blank Off On**.

6. Press **Return** > **Arb On Off** to On.

7. Press **Waveform Segments** > **Load Store** > **Waveform Utilities** > **Set Markers** > **Marker 1 2** >
   Set Marker On Range of Points.

8. Press **First Mkr Point** > **10** > **Enter**.

9. Press **Last Mkr Point** > **163830** > **Enter**.

10. Press **Apply To Waveform**.

To learn about verifying marker operation, see "To Verify Marker Operation" on page 200.

To Toggle Markers in an Existing Waveform Sequence

In a waveform sequence, you can independently toggle the operating state of the markers on each waveform segment. When you build a waveform sequence, the markers on each segment are toggled to the last marker operating state that was used.

In this example, you learn how to toggle markers within an existing waveform sequence. If you have not created waveform segments, used to build and store a waveform sequence, and configured markers for the waveform sequence, complete the steps in the previous sections, "To Create and Play Waveform Segments" on page 182, "To Build and Edit a Waveform Sequence" on page 186, and "To Place a Marker at the First Point within a Waveform Segment" on page 195.

1. Press **Mode** > **Dual ARB** > **Waveform Sequences**.

2. Highlight the desired waveform sequence (for example, TTONE+MTONE).

3. Press **Edit Selected Waveform Sequence**.

4. Highlight the desired waveform segment (for example, WFM1:TTONE).

5. Press **Toggle Markers** > **Toggle Marker 1** or **Toggle Marker 2**.

6. Highlight the next desired waveform segment.

7. Press **Toggle Marker 1** or **Toggle Marker 2**.

8. Repeat steps 6 and 7 until you have finished modifying the desired waveform segments.

9. Press **Return**.

10. Press **Name And Store**.

11. Press **Enter**.

The markers are toggled per your selections, and the changes have been saved to the selected sequence file.

An entry (1, 2, or 12) in the Mk column indicates that a marker is active. No entry in that column means that both markers are off, as shown in Figure 7-8.

Figure 7-8

To Toggle Markers As You Create a Waveform Sequence

You can combine waveform segments to create a waveform sequence while independently toggling the markers of each waveform segment.

In this example, you learn how to toggle markers while building a waveform sequence. If you have not created waveform segments, complete the steps in the previous section, "To Create and Play Waveform Segments" on page 182.

1. Press **Mode** > **Dual ARB** > **Waveform Sequences** > **Build New Waveform Sequence**.

2. Press **Insert Waveform**.

3. Highlight the desired waveform segment (for example, TTONE).

4. Press **Insert Selected Waveform** > **Insert Selected Waveform** > **Done Inserting**.

5. Highlight the first waveform segment.

   An entry (1, 2 or 12) in the Mk column indicates that a marker is active. No entry in that column means that both markers are off.

6. Press **Toggle Markers**.

7. Press **Toggle Marker 1** and **Toggle Marker 2** until only 2 is showing in the Mk column.

8. Highlight the next waveform segment.

9. Press **Toggle Marker 1** and **Toggle Marker 2** until both 1 and 2 are showing in the Mk column.

10. Press **Return**.

You now have a waveform sequence that contains two TTONE waveform segments. Marker 2 is on for the first waveform segment and markers 1 and 2 are on for the second waveform segment.

To Verify Marker Operation

In this example, you learn how to verify marker operation. If you have not created waveform segments and applied makers, complete the steps in the previous sections, "To Create and Play Waveform Segments" on page 182 and "To Place a Marker at the First Point within a Waveform Segment" on page 195.

Once you set a marker on a waveform segment, you can detect the marker pulse at the EVENT 1 or EVENT 2 connectors (EVENT 1 for this example). For more information, see "Waveform Marker Concepts" on page 201

1. Press **Mode** > **Dual ARB** > **Select Waveform**.

2. Highlight the desired waveform segment or sequence.

3. Press **ARB Off On** to On.

4. Connect an oscilloscope input to the EVENT 1 connector, and trigger on the Event 1 signal.

When a marker is present, a marker pulse is displayed on the oscilloscope.

Waveform Marker Concepts

The Dual Arb mode of the signal generator has four markers that you can place on a waveform segment. Marker 1 and Marker 2 provide auxiliary output signals to the rear-panel EVENT 1 and EVENT 2 connectors, respectively. Markers 3 and 4 are available only for custom-programmed waveforms, and they provide auxiliary output signals to pins 19 and 18 of the rear-panel AUXILIARY I/O connector, respectively. You can construct these output signals as a trigger signal to synchronize another instrument to a given portion of a waveform.

The following timing diagrams describe the effects of Markers 1 and 2 on the state of the signal at the EVENT 1 and EVENT 2 rear panel connectors.

NOTE Marker polarity selection may not be available in your version of the firmware. In this case, marker polarity is always positive.

Table 7-1 **Marker 1 and EVENT 1**

Marker File Bit 1

| Waveform point *n* | point *n*+1 | point *n*+2 | point *n*+3 | . . . |

Signal At EVENT 1 Connector

For Marker Polarity = Positive

For Marker Polarity = Negative
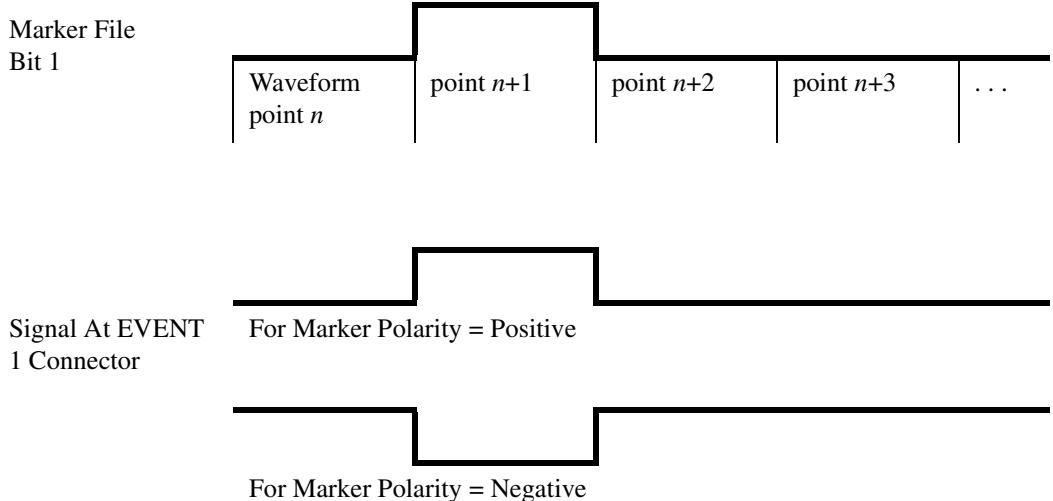
Figure 7-9

Table 7-2   Marker 2 and EVENT 2

Marker File Bit 2				
Waveform point *n*	point *n*+1	point *n*+2	point *n*+3	. . .

Signal At
EVENT 2 Connector

For Marker Polarity = Positive

For Marker Polarity = Negative

RF Output   RF Unblanked (low)

Mkr 2 to RF Blank = Off

Table 7-2   **Marker 2 and EVENT 2**

RF Unblanked

RF Output   RF Blanked   RF Blanked

Mkr 2 to RF Blank = On
Marker Polarity = Positive

RF Output   RF Unblanked   RF Unblanked

Mkr 2 to RF blank = On   RF Blanked
Marker Polarity = Negative
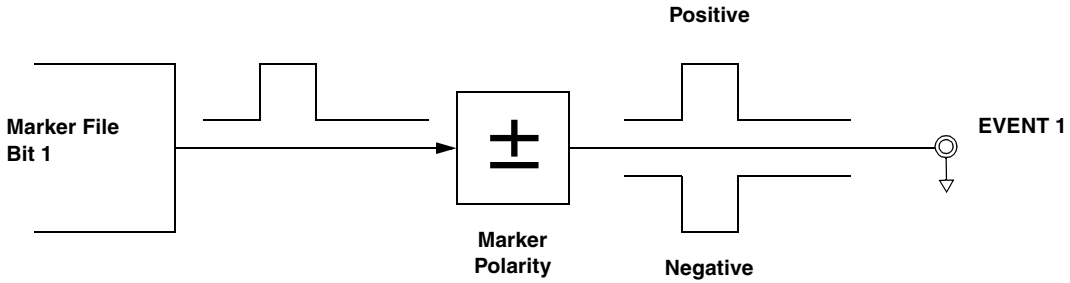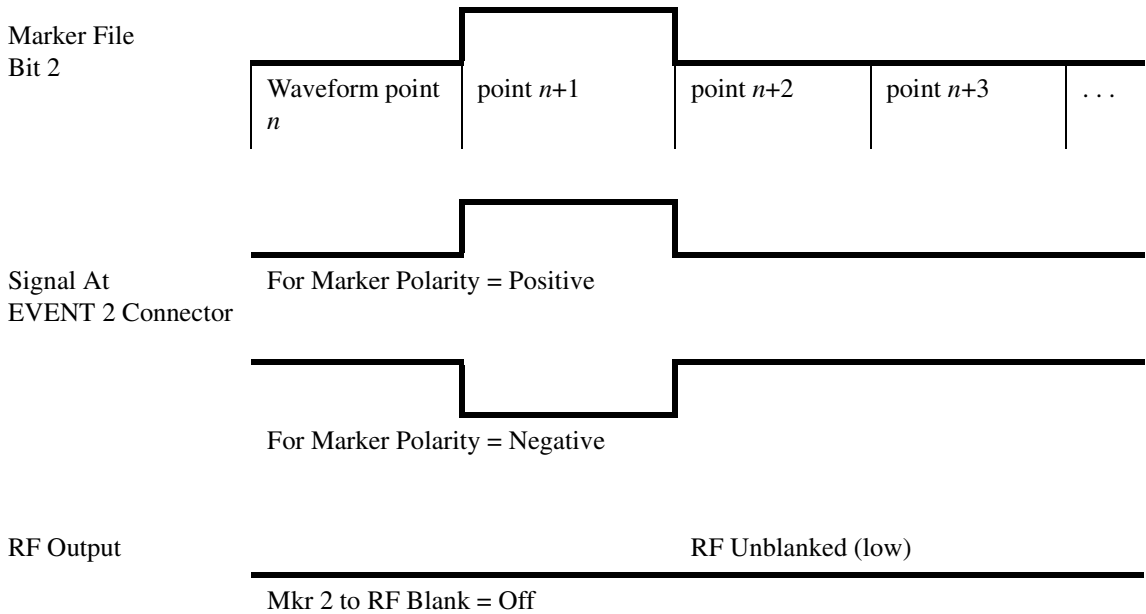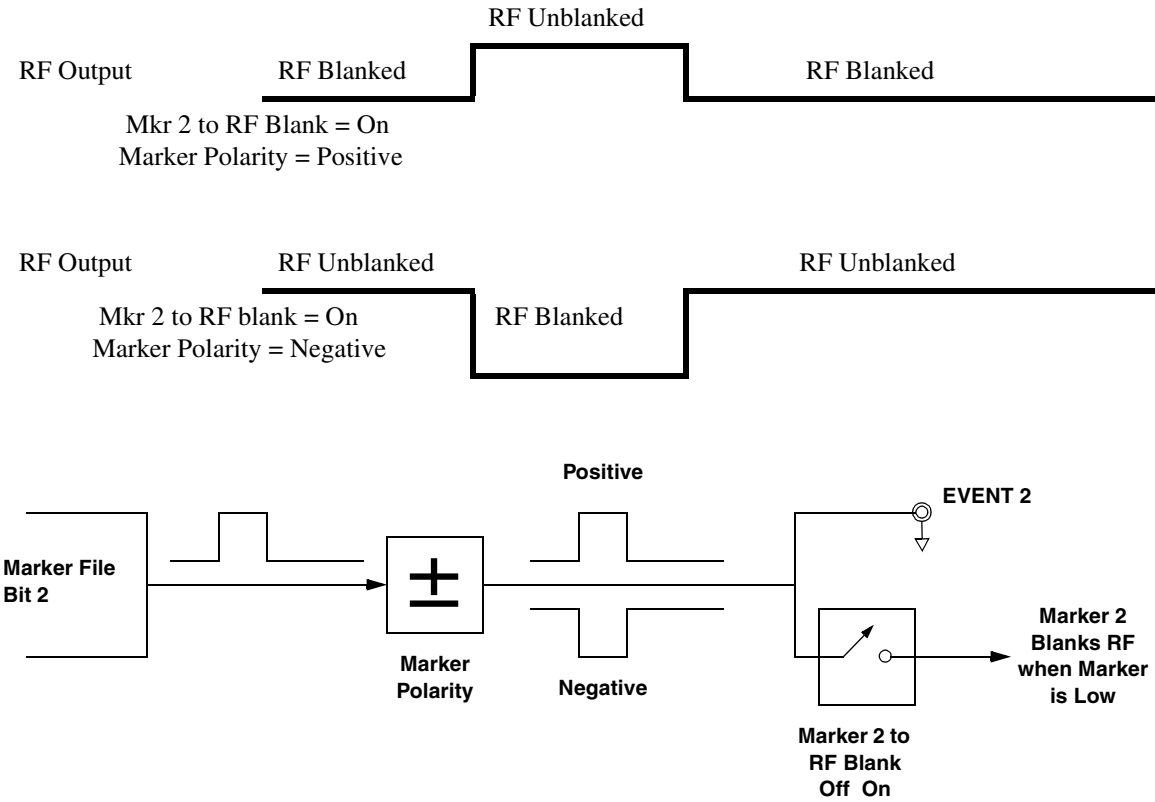
A waveform sequence comprises waveform segments. When you combine segments to form a sequence, you can enable or disable Marker 1 and/or Marker 2 on a segment-by-segment basis.

When you select a sequence to output, the markers embedded in any one segment of that sequence are output only if the sequence marker for that segment is enabled (toggled on). This makes it possible to output markers for some segments in a sequence, but not for others.
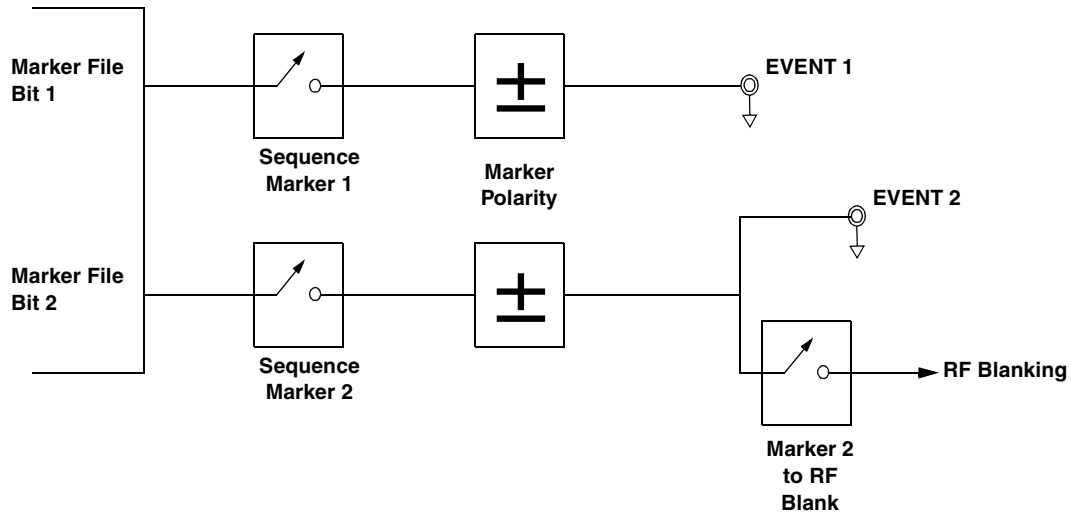
Using Waveform Triggers

The dual arbitrary waveform generator includes several different triggering options: single, gated, segment advance, and continuous. The trigger source can be the **Trigger** hardkey, a command sent through the remote interface, or an external signal applied to the TRIGGER IN rear panel connector.

To Use Segment Advance Triggering

Using this procedure, you learn how to control sequence playback of two waveform segments using segment advance triggering.

If you have not created and stored a waveform sequence, complete the steps in the previous sections, "To Create and Play Waveform Segments" on page 182, and "To Build and Edit a Waveform Sequence" on page 186.

Configuring the Waveform Sequence Trigger

1. Press **Preset**.

2. Press **Mode** > **Dual ARB** > **Select Waveform**.

3. Highlight a waveform sequence file (for example, TTONE100+MTONE200).

4. Press **Select Waveform**.

5. Press **Trigger** > **Segment Advance**.

6. Press **Trigger** > **Trigger Setup** > **Trigger Source** > **Trigger Key**.

7. Press **Return** > **Return** > **ARB Off On** to On.

   The first waveform segment in the sequence (TTONE) is now playing and modulating the RF carrier. The waveform player has been programmed to stop the playback of the current waveform segment and start the playback of the next waveform segment in the sequence when a trigger is received from the front panel **Trigger** hardkey.

   You can now enable the RF output and use your signal.

Triggering the Second Waveform

1. Press the **Trigger** hardkey.

2. Observe the second waveform segment in the sequence (MTONE) is now playing.

Pressing the **Trigger** hardkey stops the playback of the first waveform segment and starts the playback of the second waveform segment. Pressing the **Trigger** hardkey again will return the waveform player to the first waveform segment.

Programming and Downloading Waveforms

The dual arbitrary waveform generator can play waveforms that you create on a PC and download to the instrument's memory.

In this section, you will see an example of a pulse sequence that was created on a PC using Matlab, and you will learn how to download the resulting waveform file to the PSG vector signal generator's memory using Download Assistant.

Waveform files can also be downloaded by other methods, such as FTP and Intuilink. For more information on downloading files, refer to the *Programming Guide*.

To Use Matlab to Create Waveforms

Matlab is a programming tool that can be used to create specialized waveforms, such as radar and pulse sequences. Using Matlab and Agilent's integrated Download Assistant utility, you can create a single function that will build a waveform, preconfigure the waveform with playback setting information, and download the waveform to the signal generator's volatile memory for playback or sequencing.

The following Matlab M-file programming example generates and downloads a pulse pattern waveform file through the PSG vector signal generator's GPIB interface. A copy is also available on the *PSG Documentation CD-ROM* as pulsepat.m.

% Script file: pulsepat.m

%

% Purpose:

%To calculate and download an arbitrary waveform file that simulates a

%simple antenna scan pulse pattern to the PSG vector signal generator.

%

% Define Variables:

% n -- counting variable (no units)

% t -- time (seconds)

% rise -- raised cosine pulse rise-time definition (samples)

% on -- pulse on-time definition (samples)

% fall -- raised cosine pulse fall-time definition (samples)

% i -- in-phase modulation signal

% q -- quadrature modulation signal


```
n=4;                % defines the number of points in the rise-time and fall-time
t=-1:2/n:1-2/n;          % number of points translated to time
rise=(1+sin(t*pi/2))/2;    % defines the pulse rise-time shape
on=ones(1,120);           % defines the pulse on-time characteristics
fall=(1+sin(-t*pi/2))/2;   % defines the pulse fall-time shape
off=zeros(1,896);        % defines the pulse off-time characteristics


% arrange the i-samples and scale the amplitude to simulate an antenna scan
% pattern comprised of 10 pulses
i = .707*[rise on fall off...
[.9*[rise on fall off]]...
[.8*[rise on fall off]]...
[.7*[rise on fall off]]...
[.6*[rise on fall off]]...
[.5*[rise on fall off]]...
[.4*[rise on fall off]]...
[.3*[rise on fall off]]...
[.2*[rise on fall off]]...
[.1*[rise on fall off]]];


% set the q-samples to all zeroes
q = zeros(1,10240);


% define a composite iq matrix for download to the PSG using the
% PSG/ESG Download Assistant
IQData = [i + (j * q)];
```

% define a marker matrix and activate a marker to indicate the beginning of the waveform

Markers = zeros(2,length(IQData));   % fill marker array with zero, i.e no markers set

Markers(1,1) = 1;   % set marker to first point of playback

% make a new connection to the PSG over the GPIB interface

io = agt_newconnection('gpib',0,19);

% verify that communication with the PSG has been established

[status, status_description,query_result] = agt_query(io,'*idn?');

if (status < 0) return; end

% set the carrier frequency and power level on the PSG using the PSG Download Assistant

[status, status_description] = agt_sendcommand(io, 'SOURce:FREQuency 20000000000');

[status, status_description] = agt_sendcommand(io, 'POWer 0');

% define the ARB sample clock for playback

sampclk = 40000000;

% download the iq waveform to the PSG baseband generator for playback

[status, status_description] = agt_waveformload(io, IQData, 'pulsepat', sampclk, 'play', 'no_normscale', Markers);

% turn on RF output power

[status, status_description] = agt_sendcommand(io, 'OUTPut:STATe ON')

You can test your program by performing a simulated plot of the in-phase modulation signal in Matlab (see Figure 7-10 on page 210). To do this, enter `plot (i)` at the Matlab command prompt.

Figure 7-10   **Simulated Plot of In-Phase Signal**
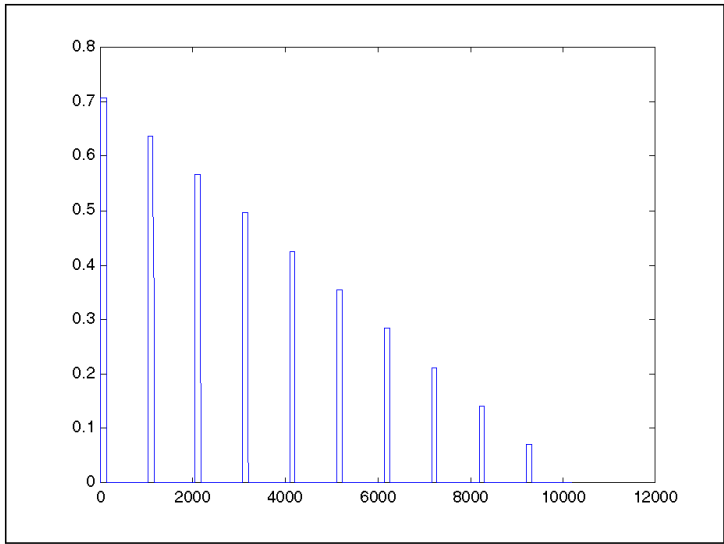
The following additional Matlab M-file programming examples are available on the
PSG Documentation CD-ROM:

barker.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a simple 7 bit barker RADAR signal to the PSG vector signal generator.

chirp.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a simple compressed pulse RADAR signal using linear FM chirp to the PSG
                    vector signal generator.

FM.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a single tone FM signal with a rate of 6 KHz, deviation of =/- 14.3 KH, Bessel
                    null of dev/rate=2.404 to the PSG vector signal generator.

nchirp.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a simple compressed pulse RADAR signal using non-linear FM chirp to the
                    PSG vector signal generator.

pulse.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a simple pulse signal to the PSG vector signal generator.

pulsedroop.m   This programming example calculates and downloads an arbitrary waveform file that
                    simulates a simple pulse signal with pulse droop to the PSG vector signal generator.

To Download Waveforms from Matlab

This procedure describes how to download a waveform file from Matlab to the PSG vector signal generator's volatile memory.

When using the Download Assistant with Matlab, the I/O interface definition and the download command are embedded in the M-file program (see the programming example in "To Use Matlab to Create Waveforms" on page 207). To download the waveform to the signal generator, execute the program in the Matlab command window by entering the name of the M-file (for example, pulsepat) at the command prompt. The Download Assistant will be instructed to download the waveform file to the signal generator.

For more information about the Download Assistant, go to www.agilent.com and search for "Download Assistant" in Test & Measurement.

NOTE	In our example, we use the GPIB interface to download waveforms. Make sure that the GPIB interface is working properly between the signal generator and the computer before downloading. See the programming guide for more information about using GPIB and other interfaces.

To Play Downloaded Waveforms

Waveform files are downloaded to the signal generator's volatile memory as WFM1 files. They can be used by the waveform player as segments or stored to non-volatile memory, just like internally created waveform files. For more information on using the waveform player, see "Using the Dual ARB Waveform Player" on page 182.

8 Multitone Waveform Generator

This chapter describes the Multitone mode, which is available only in E8267C PSG vector signal generators.

This chapter includes the following major sections:

Overview of the Multitone Waveform Generator

The multitone mode builds a waveform that has up to 64 CW signals, or tones. Using the `Multitone Setup` table editor, you can define, modify, and store waveforms for playback. Multitone waveforms are generated by the internal I/Q baseband generator.

The multitone waveform generator is typically used for testing the intermodulation distortion characteristics of multi-channel devices, such as mixers or amplifiers. Intermodulation distortion (IMD) occurs when non-linear devices with multiple input frequencies cause unwanted outputs at other frequencies or interfere with adjacent channels. The multitone waveform generator supplies a waveform with a user-specified number of tones whose IMD products can be measured using a spectrum analyzer and used as a reference when measuring the IMD generated by a device-under-test.

Multitone waveforms are created using the internal I/Q baseband generator and stored in ARB memory for playback. Although the multitone mode generates a high-quality waveform, a small amount of IMD, carrier feedthrough, and feedthrough-related IMD occurs. Carrier feedthrough may be observed when an even number of tones are generated, since there are no tones at the center carrier frequency to mask the feedthrough. To minimize carrier feedthrough for an even-numbered multitone signal, it is necessary to manually adjust the I and Q offsets while observing the center carrier frequency with a spectrum analyzer.

For measurements that require more than 64 tones or the absence of IMD and carrier feedthrough, you can create up to 1024 distortion-free multitone signals using Agilent Technologies Signal Studio software Option 408.

NOTE	For more information about multitone waveform characteristics and the PSG vector signal generator multitone personality, download *Application Note 1410* from our website by going to www.agilent.com and searching for "AN 1410" in Test & Measurement.

Creating, Viewing, and Optimizing Multitone Waveforms

This section describes how to set up, generate, and optimize a multitone waveform while viewing it with a spectrum analyzer. Although you can view a generated multitone signal using any spectrum analyzer that has sufficient frequency range, an Agilent Technologies PSA high-performance spectrum analyzer was used for this demonstration. Before generating your signal, connect the spectrum analyzer to the signal generator as shown in Figure 8-1.

Figure 8-1   **Spectrum Analyzer Setup**
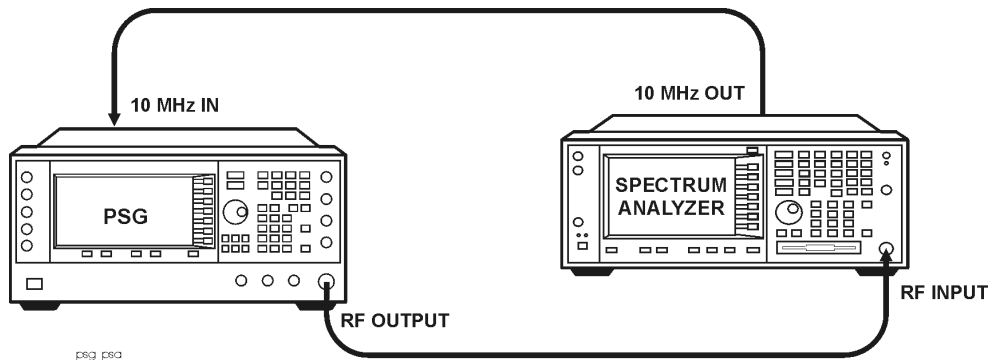
To Create a Custom Multitone Waveform

Using the `Multitone Setup` table editor, you can define, modify and store user-defined multitone waveforms. Multitone waveforms are generated by the dual arbitrary waveform generator.

1. Preset the signal generator.

2. Set the signal generator RF output frequency to 20 GHz.

3. Set the signal generator RF output amplitude to 0 dBm.

4. Press **Mode** > **Multitone** > **Initialize Table** > **Number of Tones** > **9** > **Enter**.

5. Press **Freq Spacing** > **1** > **MHz**.

6. Press **Initialize Phase Fixed Random** to Random.

7. Press **Done**.

8. Press **Multitone Off On** to On.

9. Turn on the RF output.

The multitone signal should be available at the signal generator RF OUTPUT connector.
Figure 8-2 on page 216 shows what the signal generator display should look like after all steps have been completed. Notice that the M-TONE, I/Q, RF ON, and MOD ON annunciators are displayed and the parameter settings for the signal are shown in the status area of the signal generator display. The multitone waveform is stored in volatile ARB memory.

The waveform has nine tones spaced 1 MHz apart with random initial phase values. The center tone is placed at the carrier frequency, while the other eight tones are spaced in 1 MHz increments from the center tone. If you create an even number of tones, the carrier frequency will be centered between the two middle tones.

Figure 8-2
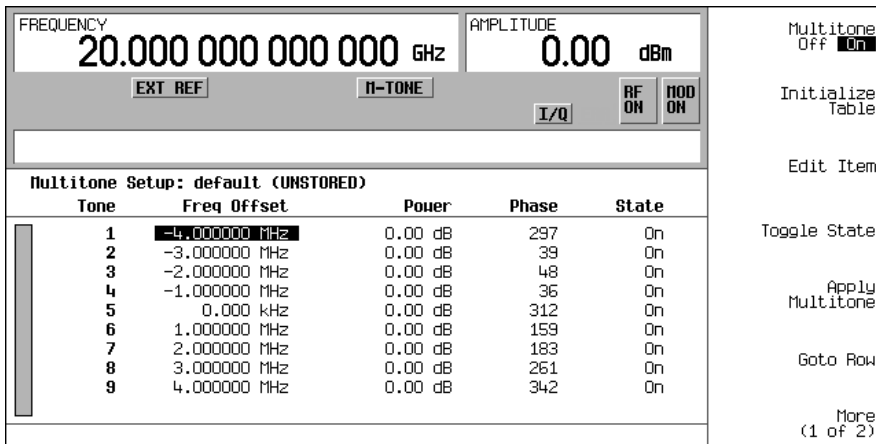
To View a Multitone Waveform

This procedure describes how to configure the spectrum analyzer to view a multitone waveform and its IMD products. Actual key presses will vary, depending on the model of spectrum analyzer you are using.

1. Preset the spectrum analyzer.

2. Set the carrier frequency to 20 GHz.

3. Set the frequency span to 20 MHz.

4. Set the amplitude for a 10 dB scale with a 4 dBm reference.

5. Adjust the resolution bandwidth to sufficiently reduce the noise floor to expose the IMD products. A 9.1 kHz setting was used in our example.

6. Turn on the peak detector.

7. Set the attenuation to 14 dB, so you're not overdriving the input mixer on the spectrum analyzer.

You should now see a waveform with nine tones and a 20 GHz center carrier frequency that is similar to the one shown in Figure 8-3 on page 217. You will also see IMD products at 1 MHz intervals above and below the highest and lowest tones.

Figure 8-3
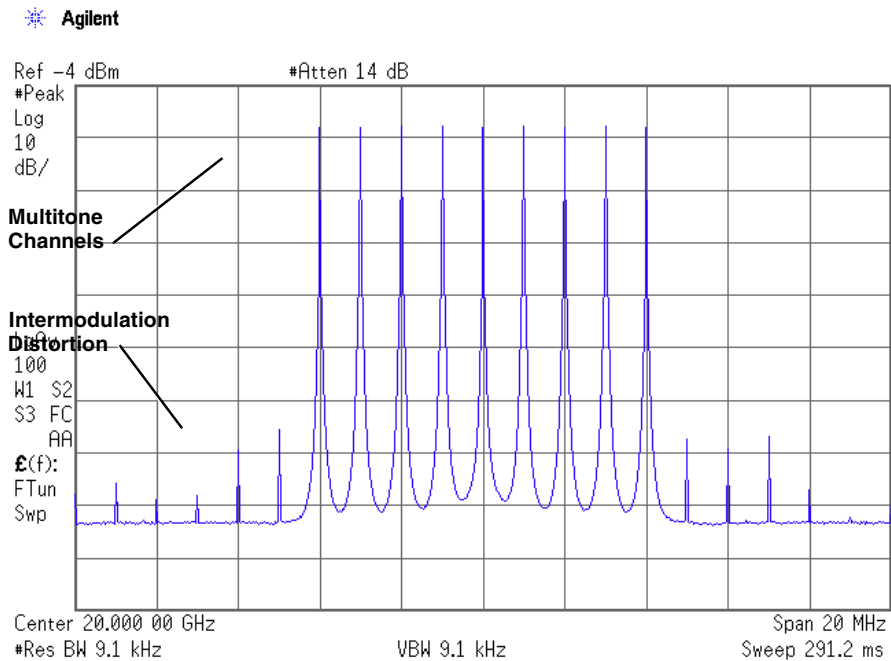
To Edit the Multitone Setup Table

This procedure builds upon the previous procedure.

1. Press **Initialize Table** > **Number of Tones** > **10** > **Enter**.

2. Press **Done**.

3. Highlight the value (On) in the State column for the tone in row 2.

4. Press **Toggle State**.

5. Highlight the value (0 dB) in the Power column for the tone in row 4.

6. Press **Edit Item** > **-10** > **dB**.

7. Highlight the value (0) in the Phase column for the tone in row 4.

8. Press **Edit Item** > **123** > **deg**.

9. Press **Apply Multitone**.

NOTE   Whenever a change is made to a setting while the multitone generator is operating (**Multitone Off On** set to On), you must apply the change by pressing the Apply Multitone softkey before the updated waveform will be generated. When you apply a change, the baseband generator creates a multitone waveform using the new settings and replaces the existing waveform in ARB memory.

You have now changed the number of tones to 10, disabled tone 2, and changed the power and phase of tone 4. Figure 8-4 on page 218 shows what the multitone setup table display on the signal generator should look like after all steps have been completed. The spectrum analyzer should display a waveform similar to the one shown in Figure 8-5 on page 219. Notice that even-numbered multitone waveforms have a small amount of carrier feedthrough at the center carrier frequency.

Figure 8-4
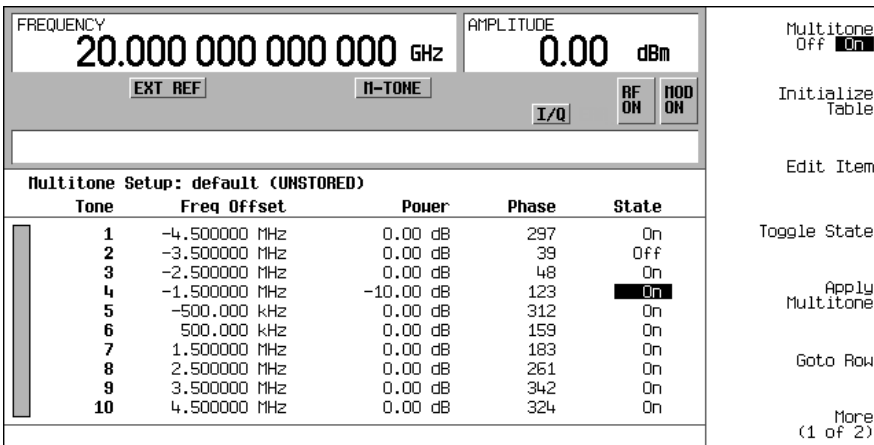
Figure 8-5
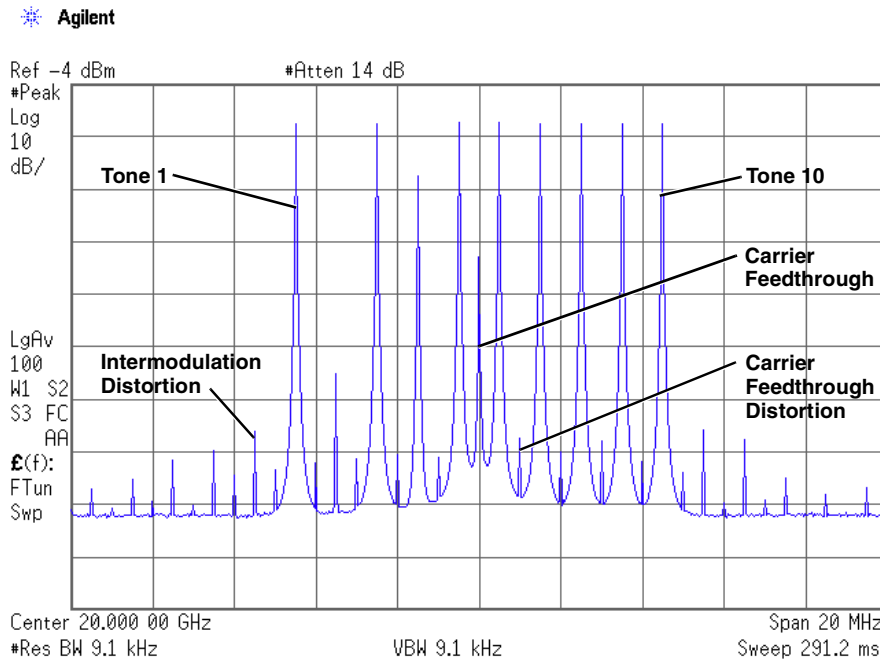
To Minimize Carrier Feedthrough

This procedure describes how to minimize carrier feedthrough and measure the difference in power between the tones and their intermodulation distortion products. Carrier feedthrough can only be observed with even-numbered multitone waveforms.

This procedure builds upon the previous procedure.

1. On the spectrum analyzer, set the resolution bandwidth for a sweep rate of about 100-200 ms. This will allow you to dynamically view the carrier feedthrough spike as you make adjustments.

2. On the signal generator, press **I/Q** > **I/Q Adjustments** > I/Q Adjustments Off On to On.

3. Press I Offset and turn the rotary knob while observing the carrier feedthrough with the spectrum analyzer. Changing the I offset in the proper direction will reduce the feedthrough level. Adjust the level as low as possible.

4. Press Q Offset and turn the rotary knob to further reduce the carrier feedthrough level.

5. Repeat steps 3 and 4 until you have reached the lowest possible carrier feedthrough level.

6. On the spectrum analyzer, return the resolution bandwidth to its previous setting.

7. Turn on waveform averaging.

8. Create a marker and place it on the peak of one of the end tones.

9. Create a delta marker and place it on the peak of the adjacent intermodulation product, which should be spaced 10 MHz from the marked tone.

10. Measure the power difference between the tone and its distortion product.

You should now see a display that is similar to the one shown in . Your optimized multitone signal can now be used to measure the IMD products generated by a device-under-test.

Note that carrier feedthrough changes with time and temperature. Therefore, you will need to periodically readjust your I and Q offsets to keep the signal optimized.

Figure 8-6
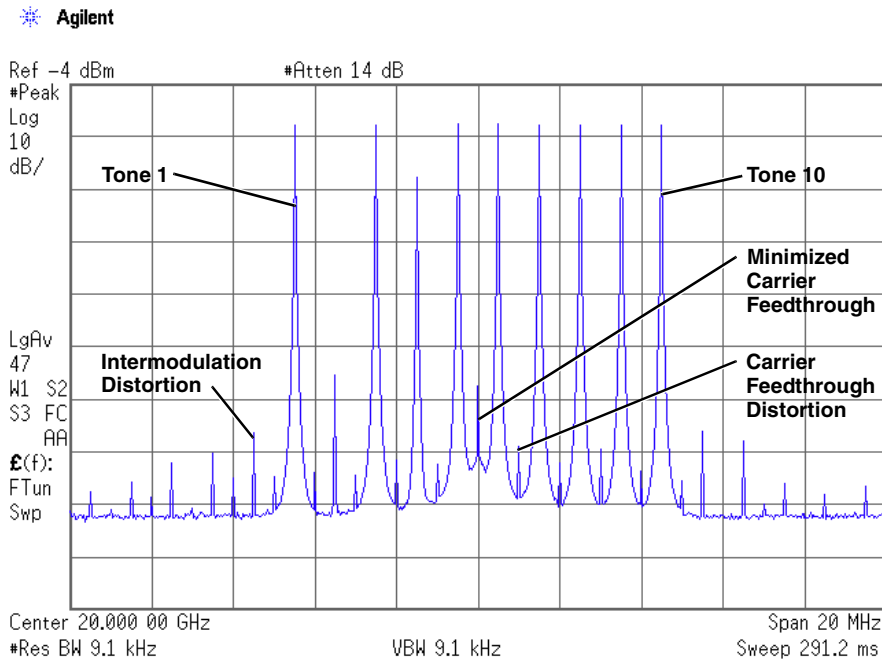
To Determine Peak to Average Characteristics

This procedure describes how to set the phases of the tones in a multitone waveform and determine the peak to average characteristics by plotting the complementary cumulative distribution function (CCDF).

1. Press **Mode > Multitone > Initialize Table > Number of Tones > 64 > Enter**.

2. Press **Freq Spacing > 20 > kHz**.

3. Press **Initialize Phase Fixed Random** to Fixed.

4. Press **Done**.

5. Press **Apply Multitone**.

6. Press **More (1 of 2) > Waveform Statistics > Plot CCDF**.

   You should now see a display that is similar to the one shown in Figure 8-7. The CCDF plot displays the peak to average characteristics of the waveform with all phases set to zero.

Figure 8-7   CCDF Plot with Fixed Phase Set

Peak
Power
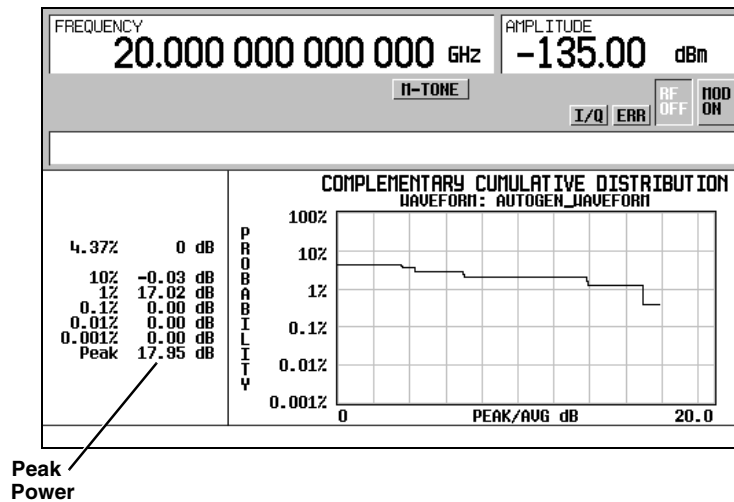
7. Press **Mode Setup > Initialize Table**.

8. Press **Initialize Phase Fixed Random** to Random.

9. Press **Random Seed Fixed Random** to Random.

10. Press **Done**.

11. Press **Apply Multitone**.

12. Press **More (1 of 2) > Waveform Statistics > Plot CCDF**.

You should now see a display that is similar to the one shown in Figure 8-8. The CCDF plot displays the peak to average characteristics of the waveform with randomly generated phases and a random seed.

The random phase setup simulates the typically random nature of multitone waveforms. Notice that randomly distributed phases result in a much lower peak to average ratio than fixed phases. An increase in the number of tones with random phases will decrease the probability of a maximum peak power occurrence.

Figure 8-8   **CCDF Plot with Random Phase Set**

Peak
Power
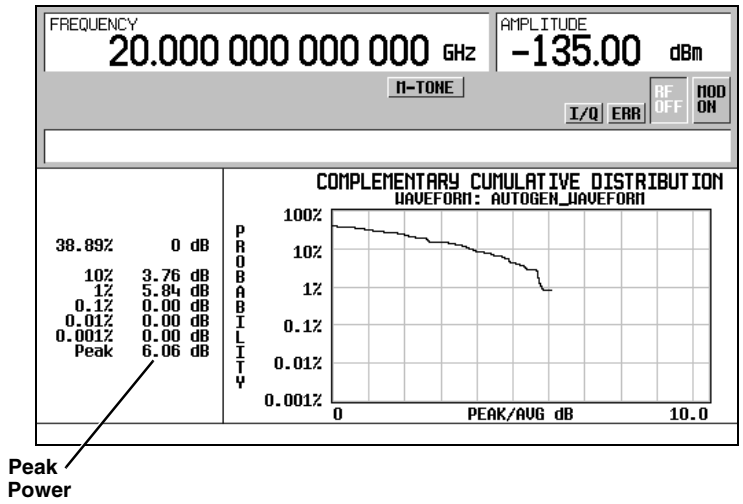
9 Two-Tone Waveform Generator

This chapter describes the Two Tone mode, which is available only in E8267C PSG vector signal generators.

This chapter includes the following major sections:

- *"Overview of the Two-Tone Waveform Generator" on page 224*
- *"Creating, Viewing, and Modifying Two-Tone Waveforms" on page 225*

Overview of the Two-Tone Waveform Generator

The two-tone mode builds a waveform that has two equal-powered CW signals, or tones. The default waveform has two tones that are symmetrically spaced from the center carrier frequency, and have user-defined amplitude, carrier frequency, and frequency separation settings. The user can also align the tones left or right, relative to the carrier frequency.

The two-tone waveform generator is designed for testing the intermodulation distortion characteristics of non-linear devices, such as mixers or amplifiers. Intermodulation distortion (IMD) occurs when non-linear devices with multiple input frequencies interfere with adjacent channels or cause unwanted outputs at other frequencies. The two-tone waveform generator supplies a signal whose IMD products can be measured using a spectrum analyzer and used as a reference when measuring the IMD generated by a device-under-test.

Two-tone waveforms are created using the internal I/Q baseband generator and stored in ARB memory for playback. Although the two-tone mode generates a high-quality waveform, a small amount of IMD occurs. In addition to IMD, a small amount of carrier feedthrough and feedthrough-related IMD may be present when the spacing between the tones is centered on the carrier frequency. To minimize carrier feedthrough for a two-tone signal, it is necessary to manually adjust the I and Q offsets while observing the center carrier frequency with a spectrum analyzer. For measurements that require the absence of IMD and carrier feedthrough, you can create distortion-free multitone signals using Agilent Technologies Signal Studio software Option 408.

NOTE	For more information about two-tone waveform characteristics and the PSG vector signal generator two-tone personality, download *Application Note 1410* from our website by going to www.agilent.com and searching for "AN 1410" in Test & Measurement.

Creating, Viewing, and Modifying Two-Tone Waveforms

This section describes how to set up, generate, and modify a two-tone waveform while viewing it with a spectrum analyzer. Although you can view a generated two-tone signal using any spectrum analyzer that has sufficient frequency range, an Agilent Technologies PSA Series High-Performance Spectrum Analyzer was used for this demonstration. Before generating your signal, connect the spectrum analyzer to the signal generator as shown in Figure 9-1.

Figure 9-1   **Spectrum Analyzer Setup**
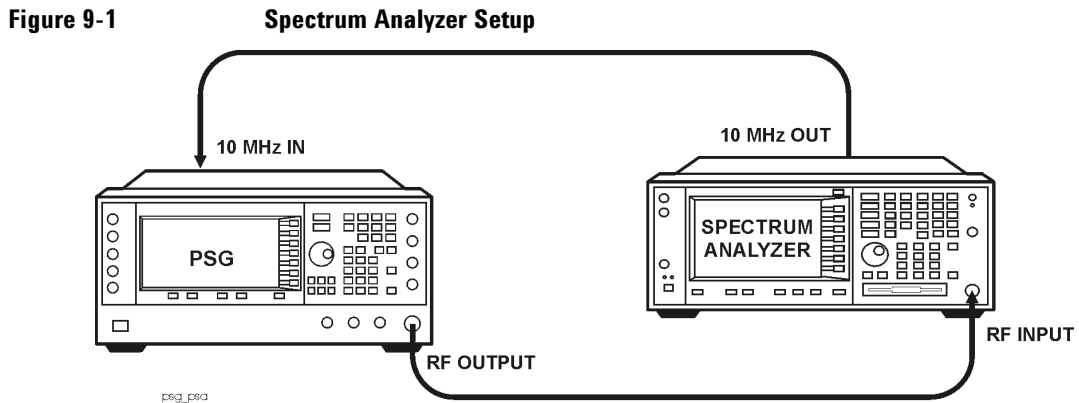
To Create a Two-Tone Waveform

This procedure describes how to create and a basic, center-aligned, two-tone waveform.

1. Preset the signal generator.

2. Set the signal generator RF output frequency to 20 GHz.

3. Set the signal generator RF output amplitude to 0 dBm.

4. Press **Mode** > **Two Tone** > **Freq Separation** > **10** > **MHz**.

5. Press **Two Tone Off On** to On.

6. Turn on the RF output.

The two-tone signal is now available at the signal generator RF OUTPUT connector. Figure 9-2 shows what the signal generator display should look like after all steps have been completed. Notice that the T-TONE, I/Q, RF ON, and MOD ON annunciators are displayed and the parameter settings for the signal are shown in the status area of the signal generator display.
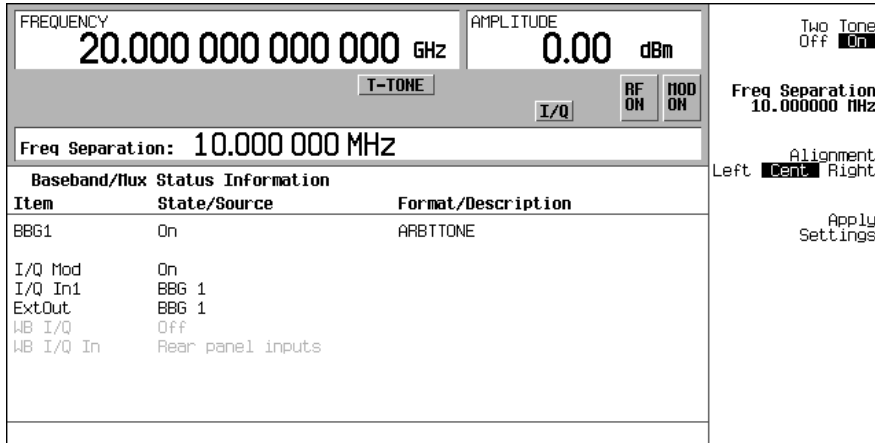
Figure 9-2

To View a Two-Tone Waveform

This procedure describes how to configure the spectrum analyzer to view a two-tone waveform and its IMD products. Actual key presses will vary, depending on the model of spectrum analyzer you are using.

1. Preset the spectrum analyzer.

2. Set the carrier frequency to 20 GHz.

3. Set the frequency span to 60 MHz.

4. Set the amplitude for a 10 dB scale with a 4 dBm reference.

5. Adjust the resolution bandwidth to sufficiently reduce the noise floor to expose the IMD products. A 9.1 kHz setting was used in our example.

6. Turn on the peak detector.

7. Set the attenuation to 14 dB, so you're not overdriving the input mixer on the spectrum analyzer.

You should now see a two-tone waveform with a 20 GHz center carrier frequency that is similar to the one shown in Figure 9-3 on page 227. You will also see IMD products at 10 MHz intervals above and below the generated tones, and a carrier feedthrough spike at the center frequency with carrier feedthrough distortion products at 10 MHz intervals above and below the center carrier frequency.

Figure 9-3
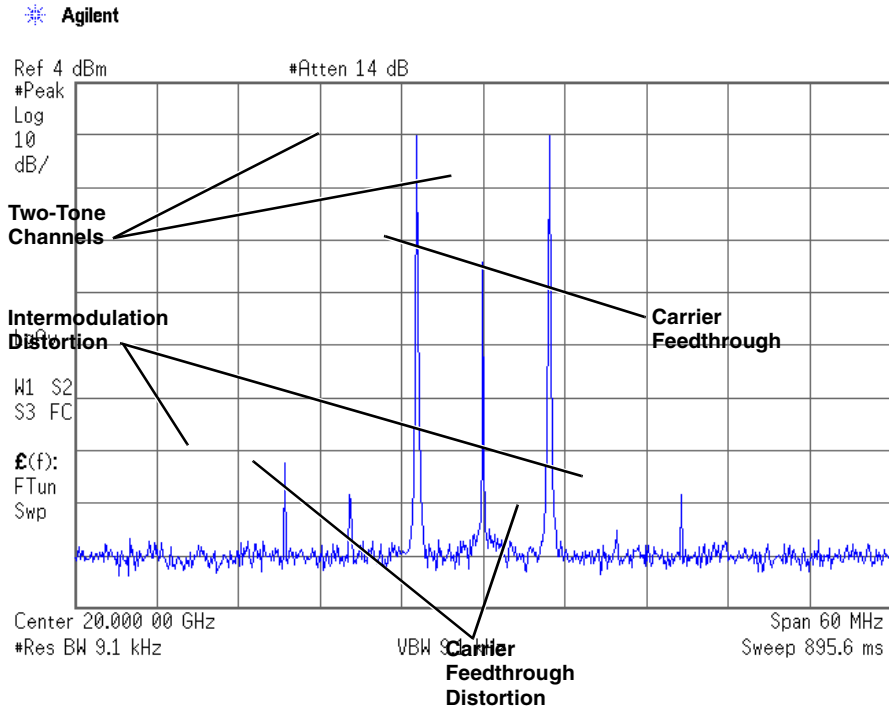
To Minimize Carrier Feedthrough

This procedure describes how to minimize carrier feedthrough and measure the difference in power between the tones and their intermodulation distortion products. Carrier feedthrough only occurs with center-aligned two-tone waveforms.

This procedure builds upon the previous procedure.

1. On the spectrum analyzer, set the resolution bandwidth for a sweep rate of about 100-200 ms. This will allow you to dynamically view the carrier feedthrough spike as you make adjustments.

2. On the signal generator, press **I/Q > I/Q Adjustments** > I/Q Adjustments Off On to On.

3. Press I Offset and turn the rotary knob while observing the carrier feedthrough with the spectrum analyzer. Changing the I offset in the proper direction will reduce the feedthrough level. Adjust the level as low as possible.

4. Press Q Offset and turn the rotary knob to further reduce the carrier feedthrough level.

5. Repeat steps 3 and 4 until you have reached the lowest possible carrier feedthrough level.

6. On the spectrum analyzer, return the resolution bandwidth to its previous setting.

7. Turn on waveform averaging.

8. Create a marker and place it on the peak of one of the two tones.

9. Create a delta marker and place it on the peak of the adjacent intermodulation product, which should be spaced 10 MHz from the marked tone.

10. Measure the power difference between the tone and its distortion product.

You should now see a display that is similar to the one shown in Figure 9-4 on page 228. Your optimized two-tone signal can now be used to measure the IMD products generated by a device-under-test.

Note that carrier feedthrough changes with time and temperature. Therefore, you will need to periodically readjust your I and Q offsets to keep your signal optimized.

Figure 9-4
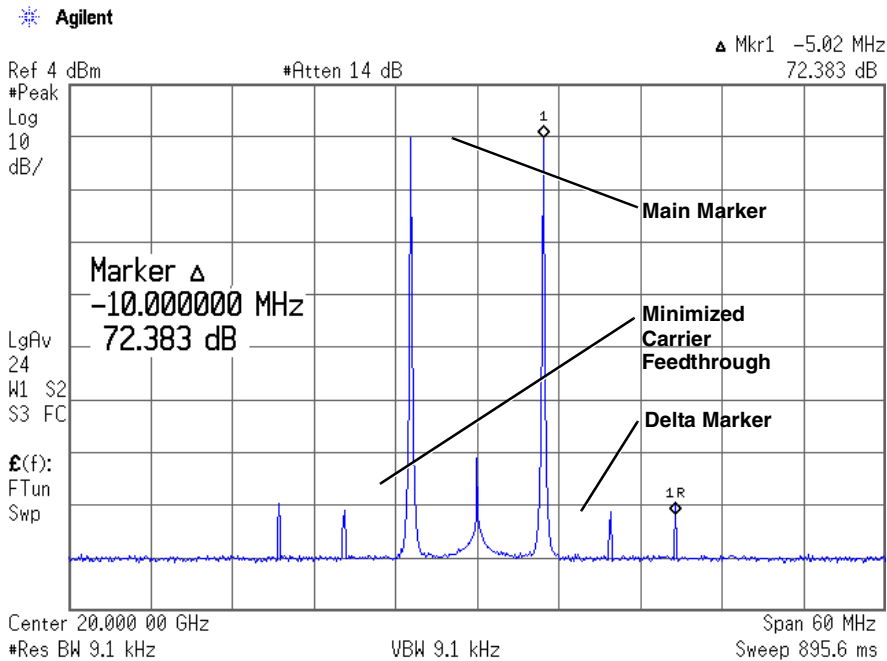
To Change the Alignment of a Two-Tone Waveform

This procedure describes how to align a two-tone waveform left or right, relative to the center carrier frequency. Left or right alignment eliminates carrier feedthrough, since the frequency of one of the tones is the same as the carrier frequency. However, image frequency interference caused by left or right alignment may cause minor distortion of the two-tone signal.

This procedure builds upon the previous procedure.

1. On the signal generator, press **Mode Setup** > **Alignment Left Cent Right** to Left.

2. Press **Apply Settings** to regenerate the waveform.

NOTE   Whenever a change is made to a setting while the two-tone generator is operating (**Two Tone Off On** set to On), you must apply the change by pressing the Apply Settings softkey before the updated waveform will be generated. When you apply a change, the baseband generator creates a two-tone waveform using the new settings and replaces the existing waveform in ARB memory.

---

3. On the spectrum analyzer, temporarily turn off waveform averaging to refresh your view more quickly.

You should now see a left-aligned two-tone waveform that is similar to the one shown in .

Figure 9-5
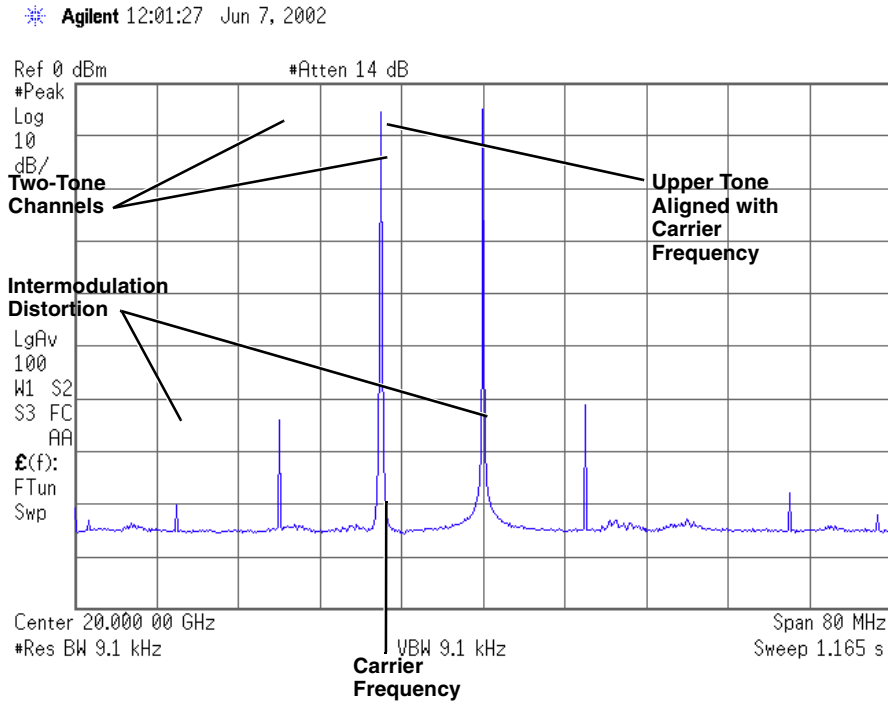
10 Troubleshooting

This chapter provides troubleshooting information for Agilent PSG signal generators.

This chapter includes the following major sections:

- "If You Encounter a Problem" on page 232
- "Basic Signal Generator Operations" on page 233
- "Signal Generator Lock-Up" on page 241
- "Upgrading Firmware" on page 243
- "Returning a Signal Generator to Agilent Technologies" on page 244

If You Encounter a Problem

If the signal generator is not operating properly, refer to the proper section in this chapter for a possible solution. If you do not find a solution, refer to the *Service Guide*.

NOTE	If the signal generator displays errors, always read the error message text by pressing **Utility** > **Error Info**.

Basic Signal Generator Operations

Cannot Turn Off Help Mode

1. Press **Utility** > **Instrument Info/Help Mode**

2. Press **Help Mode Single Cont** until Single is highlighted.

The signal generator has two help modes; single and continuous.

When you press **Help** in single mode (the factory preset condition), help text is provided for the next key you press. Pressing another key will exit the help mode and activate the key's function.

When you press **Help** in continuous mode, help text is provided for the next key you press and that key's function is also activated (except for **Preset**). You will stay in help mode until you press **Help** again or change to single mode.

No RF Output

Check the RF ON/OFF annunciator on the display. If it reads RF OFF, press **RF On/Off** to toggle the RF output on.

The Power Supply has Shut Down

If the power supply is not working, it requires repair or replacement. There is no user-replaceable power supply fuse. Refer to the Service Guide for instructions.

Signal Loss While Working with Mixers

If you experience signal loss at the signal generator's RF output during low-amplitude coupled operation with a mixer, you can solve the problem by adding attenuation and increasing the RF output amplitude of the signal generator.

Figure 10-1 on page 234 shows a hypothetical configuration in which the signal generator provides a low amplitude signal to a mixer.

Figure 10-1   **Effects of Reverse Power on ALC**
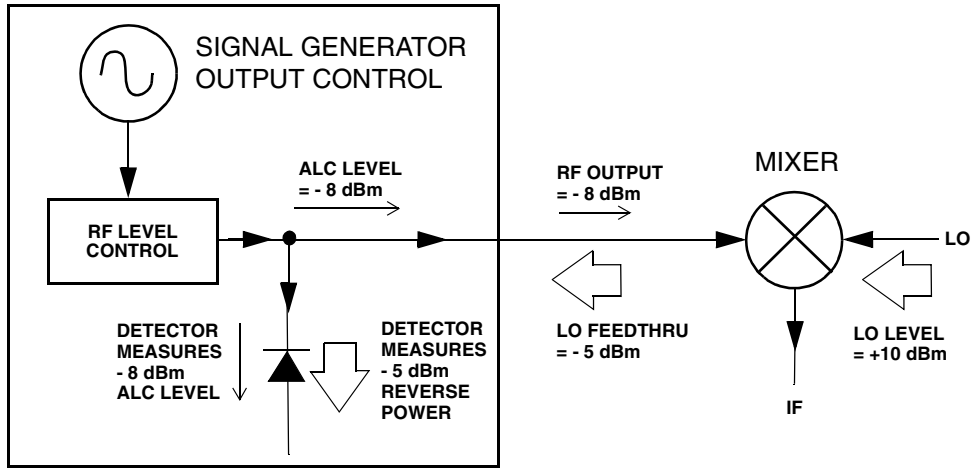
The internally leveled signal generator RF output (and ALC level) is -8 dBm. The mixer is driven with an LO of +10 dBm and has an LO-to-RF isolation of 15 dB. The resulting LO feedthrough of -5 dBm enters the signal generator's RF output connector and arrives at the internal detector.

Depending on frequency, it is possible for most of this LO feedthrough energy to enter the detector. Since the detector responds to its total input power regardless of frequency, this excess energy causes the ALC to reduce the RF output of the signal generator. In this example, the reverse power across the detector is actually greater than the ALC level, which may result in loss of signal at the RF output.

Figure 10-2 on page 235 shows a similar configuration with the addition of a 10 dB attenuator connected between the RF output of the signal generator and the input of the mixer. The signal generator's ALC level is increased to +2 dBm and transmitted through a 10 dB attenuator to achieve the required -8 dBm amplitude at the mixer input.

Figure 10-2   Reverse Power Solution
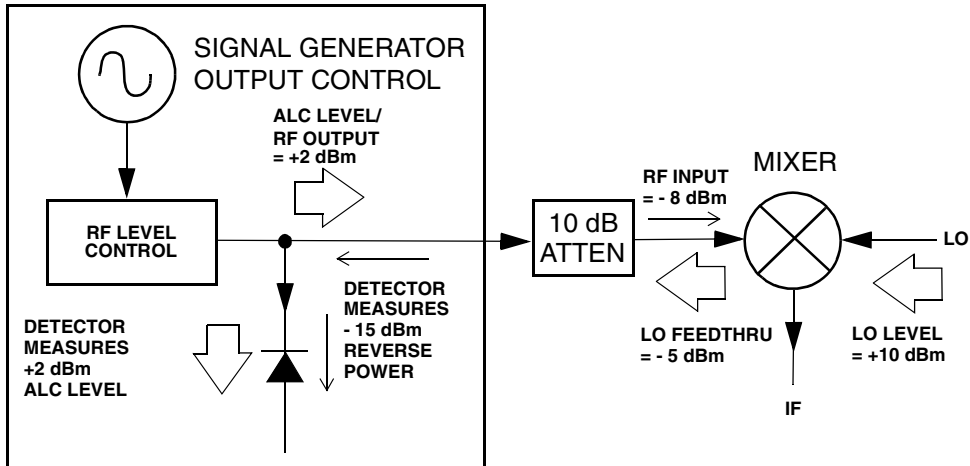
Compared to the original configuration, the ALC level is 10 dB higher while the attenuator reduces the LO feedthrough (and the RF output of the signal generator) by 10 dB. Using the attenuated configuration, the detector is exposed to a +2 dBm desired signal versus the
-15 dBm undesired LO feedthrough. This 17 dB difference between desired and undesired energy results in a maximum 0.1 dB shift in the signal generator's RF output level.

Signal Loss While Working with Spectrum Analyzers

The effects of reverse power can cause problems with the signal generator's RF output when the signal generator is used with a spectrum analyzer that does not have preselection capability.

Some spectrum analyzers have as much as +5 dBm LO feedthrough at their RF input port at some frequencies. If the frequency difference between the LO feedthrough and the RF carrier is less than the ALC bandwidth, the LO's reverse power can cause amplitude modulation of the signal generator's RF output. The rate of the undesired AM equals the difference in frequency between the spectrum analyzer's LO feedthrough and the RF carrier of the signal generator.

Reverse power problems can be solved by using one of two unleveled operating modes: ALC off or power search.

ALC Off Mode

ALC off mode deactivates the automatic leveling circuitry prior to the signal generator's RF output. In this mode, a power meter is required to measure the output of the signal generator and assist in achieving the required output power at the point of detection.

To set the signal generator to the ALC off mode, follow these steps:

1. Press **Preset**.

2. Press **Frequency**, enter the required frequency, and terminate the entry with the appropriate terminator softkey.

3. Press **Amplitude**, enter the required amplitude, and terminate the entry with the appropriate terminator softkey.

4. Set **RF On/Off** to Off

5. Press **Amplitude** > **ALC Off On** to Off.

   This deactivates the signal generator's automatic leveling control.

6. Monitor the RF output amplitude as measured by the power meter.

7. Press **Amplitude** and adjust the signal generator's RF output amplitude until the desired power is measured by the power meter.

Power Search Mode

Power search mode executes a power search routine that temporarily activates the ALC, calibrates the power of the current RF output, and then disconnects the ALC circuitry.

To set the signal generator to manual fixed power search mode, follow these steps:

1. Press **Preset**.

2. Press **Frequency**, enter the required frequency, and terminate the entry with the appropriate terminator softkey.

3. Press **Amplitude**, enter the required amplitude, and terminate the entry with the appropriate terminator softkey.

4. Set **ALC Off On** to Off.

   This deactivates the ALC circuitry.

5. Set **RF On/Off** to On.

6. Press **Do Power Search**.

   This executes the manual fixed power search routine, which is the default mode.

There are three power search modes: manual, automatic, and span.

When **Power Search** is set to Manual, pressing **Do Power Search** executes the power search calibration routine for the current RF frequency and amplitude. In this mode, if there is a change in RF frequency or amplitude, you will need to press **Do Power Search** again.

When **Power Search** is set to Auto, the calibration routine is executed whenever the frequency or amplitude of the RF output is changed.

When **Power Search** is set to Span, pressing **Do Power Search** executes the power search calibration routine over a selected range of frequencies at one time. The power search corrections are then stored and used whenever the signal generator is tuned within the selected range of frequencies.

RF Output Power too Low

1. Look for an OFFS or REF indicator in the AMPLITUDE area of the display.

   OFFS tells you that an amplitude offset has been set. An amplitude offset changes the value shown in the AMPLITUDE area of the display but does not affect the output power. The amplitude displayed is equal to the current power output by the signal generator hardware plus the value for the offset.

   To eliminate the offset, press the following keys:

   Amplitude > **More (1 of 2)** > **Ampl Offset** > **0** > **dB**.

   REF tells you that the amplitude reference mode is activated. When this mode is on, the displayed amplitude value is *not* the output power level. It is the current power output by the signal generator hardware minus the reference value set by the **Ampl Ref Set** softkey.

   To exit the reference mode, follow these steps:

   a. Press **Amplitude** > **More (1 of 2)**.

   b. Press **Ampl Ref Off On** until Off is highlighted.

   You can then reset the output power to the desired level.

2. If you are using the signal generator with an external mixer, see "Signal Loss While Working with Mixers" on page 233.

3. If you are using the signal generator with a spectrum analyzer, see "Signal Loss While Working with Spectrum Analyzers" on page 235.

No Modulation at the RF Output

Check the MOD ON/OFF annunciator on the display. If it reads MOD OFF, press **Mod On/Off** to toggle the modulation on.

Although you can set up and enable various modulations, the RF carrier is modulated only when you have also set **Mod On/Off** to On.

For digital modulation, make sure that **I/Q Off On** is set to On.

Sweep Appears to be Stalled

The current status of the sweep is indicated as a shaded rectangle in the progress bar. You can observe the progress bar to determine if the sweep is progressing. If the sweep appears to have stalled, check the following:

❏ Have you turned on the sweep by pressing any of the following key sequences?

<div align="center">

Sweep/List > **Sweep** > **Freq**

Sweep/List > **Sweep** > **Ampl**

Sweep/List > **Sweep** > **Freq & Ampl**

</div>

❏ Is the sweep in continuous mode? If the sweep is in single mode, be sure that you have pressed the **Single Sweep** softkey at least once since completion of the prior sweep. Try setting the mode to continuous to determine if the missing single sweep is blocking the sweep.

❏ Is the signal generator receiving the appropriate sweep trigger? Try setting the **Sweep Trigger** softkey to Free Run to determine if a missing sweep trigger is blocking the sweep.

❏ Is the signal generator receiving the appropriate point trigger? Try setting the **Point Trigger** softey to Free Run to determine if a missing point trigger is blocking the sweep.

❏ Is the dwell time appropriate? Try setting the dwell time to one second to determine if the dwell time was set to a value which was too slow or too fast for you to see.

❏ Do you have at least two points in your step sweep or list sweep?

Cannot Turn Off Sweep Mode

Press **Sweep/List** > **Sweep** > **Off**.

In the sweep mode menu you can choose to set the sweep to various sweep types or to turn sweep off.

Incorrect List Sweep Dwell Time

If the signal generator does not dwell for the correct period of time at each sweep list point, follow these steps:

1. Press **Sweep/List > Configure List Sweep**.

   This displays the sweep list values.

2. Check the sweep list dwell values for accuracy.

3. Edit the dwell values if they are incorrect.

NOTE   The effective dwell time at the RF OUTPUT connector is the sum of the value set for the dwell plus processing time, switching time, and settling time. This additional time added to the dwell is generally a few milliseconds. The TTL/CMOS output available at the TRIG OUT connector, however, is asserted high only during the actual dwell time.

   If the list dwell values are correct, continue to the next step.

4. Observe if the **Dwell Type List Step** softkey is set to Step.

   When Step is selected, the signal generator will sweep the list points using the dwell time set for step sweep rather than the sweep list dwell values.

   To view the step sweep dwell time, follow these steps:

   a. Press **Configure Step Sweep**.

   b. Observe the value set for the **Step Dwell** softkey.

List Sweep Information is Missing from a Recalled Register

List sweep information is not stored as part of the instrument state in an instrument state register. Only the current list sweep is available to the signal generator. List sweep data can be stored in the instrument catalog. For instructions, see "Storing Files to the Memory Catalog" on page 67.

Data Storage

Registers With Previously Stored Instrument States are Empty

The save/recall registers are backed-up by a battery when line power to the signal generator is not connected. The battery may need to be replaced.

To verify that the battery has failed, follow these steps:

1. Turn off line power to the signal generator.

2. Unplug the signal generator from line power.

3. Plug in the signal generator.

4. Turn on the signal generator.

5. Observe the display for error messages.

    If either error message −311 or −700 is stored in the error message queue, the signal generator's battery has failed.

6. Refer to the *Service Guide* for battery replacement instructions.

Saved an Instrument State in a Register but the Register is Empty or Contains the Wrong State

If you have selected a register number that is greater than 99, the signal generator will automatically select register 99 to save your instrument state.

If the register number you intended to use is empty or contains the wrong instrument state, press the following keys:

Recall > **99** > **Enter**.

This recalls register 99. The lost instrument state may be saved there.

Signal Generator Lock-Up

If the signal generator is locked up, check the following:

- Make sure that the signal generator is not in remote mode. (In remote mode, the R annunciator will appear on the display.) Press **Local** to exit remote mode and unlock the front panel keypad.

- Make sure that the signal generator is not in local lockout condition. Local lockout will prevent front panel operation of the signal generator. For more information on local lockout, refer to the *Programming Guide*.

- Check for a progress bar on the signal generator display which indicates that an operation is in progress.

- Press **Preset**.

- Cycle power on the signal generator.

Fail-Safe Recovery Sequence

The fail-safe recovery sequence should only be used if the previous suggestions do not resolve the problem.

NOTE   This process will reset the signal generator, but it will destroy data.

The fail-safe recovery sequence will destroy the following types of data:

- all user files (instrument state and data files)
- DCFM/DCΦM calibration data
- persistent states

Do not attempt to perform any other front panel or remote operations during the fail-safe sequence.

To run the fail-safe sequence, follow these steps:

1. Hold down the **Preset** key while cycling power.

2. Continue to hold down the **Preset** key until the following message is displayed:

WARNING   **You are entering the diagnostics menu which can cause unpredictable instrument behavior. Are you sure you want to continue?**

| CAUTION | Carefully read the entire message! It may list additional risks with this procedure. |

3. Release the **Preset** key.

4. Press **Continue** to continue with the sequence (or **Abort** to abort with no lost files).

At the conclusion of the sequence, follow these steps:

1. Cycle power.

   Cycling power restores all previously installed options. You should expect to see several error messages resulting from calibration files being restored from EEPROM.

2. Perform the DCFM/DCΦM calibration.

   Refer to the **DCFM/DCΦM Cal** softkey description in the Key and Data Field Reference Volume 1.

3. Agilent Technologies is interested in the circumstances that made it necessary for you to initiate this procedure. Please contact us at the appropriate telephone number listed in Table 10-1 on page 244. We would like to help you eliminate any repeat occurrences.

Upgrading Firmware

The firmware in your signal generator may be upgraded when new firmware is released. New firmware releases may contain signal generator features and functionality not available in previous firmware releases.

To inquire about the availability of new signal generator firmware, contact Agilent at `http://www.agilent.com/find/upgradeassistant`, or call the appropriate number listed in Table 10-1 on page 244.

Returning a Signal Generator to Agilent Technologies

To return your signal generator to Agilent Technologies, follow these steps:

1. Be prepared to give your service representative as much information as possible regarding the signal generator's problem.

2. Call the phone number listed in Table 10-1 appropriate to the signal generator's location. After sharing information regarding the signal generator and its condition, you will receive information regarding where to ship your instrument for repair.

3. Ship the signal generator in the original factory packaging materials, if they are available. If not, use similar packaging to properly protect the instrument.

Table 10-1   **Contacting Agilent**

Online assistance: www.agilent.com/find/assist

United States	**Latin America**	**Canada**	**Europe**
(tel) 1 800 452 4844	(tel) (305) 269 7500	(tel) 1 877 894 4414	(tel) (+31) 20 547 2323
	(fax) (305) 269 7599	(fax) (905) 282-6495	(fax) (+31) 20 547 2390

New Zealand	**Japan**	**Australia**	
(tel) 0 800 738 378	(tel) (+81) 426 56 7832	(tel) 1 800 629 485	
(fax) (+64) 4 495 8950	(fax) (+81) 426 56 7840	(fax) (+61) 3 9210 5947	

Asia Call Center Numbers

Country	Phone Number	Fax Number
Singapore	1-800-375-8100	(65) 836-0252
Malaysia	1-800-828-848	1-800-801664
Philippines	(632) 8426802 1-800-16510170 (PLDT Subscriber Only)	(632) 8426809 1-800-16510288 (PLDT Subscriber Only)
Thailand	(088) 226-008 (outside Bangkok) (662) 661-3999 (within Bangkok)	(66) 1-661-3714
Hong Kong	800-930-871	(852) 2506 9233
Taiwan	0800-047-866	(886) 2 25456723
People's Republic of China	800-810-0189 (preferred) 10800-650-0021	10800-650-0121
India	1-600-11-2929	000-800-650-1101

Index

B

baseband generator options, 9
basic operation, 33–72
BBG DATA CLOCK
  setting to external or internal, 169
BBG reference
  setting external frequency, 168
  setting to external or internal, 168
BbT adjustment, 112, 141
binary, 66
Bit, 66
bits per symbol, equation, 172
burst shape
  configuring fall delay, 165
  configuring fall time, 165
  configuring rise delay, 165
  configuring rise time, 165
  Real Time I/Q, 163
  recalling from Memory Catalog, 167
  using user defined, 165

C

carrier feedthrough, minimizing, 219, 221, 227
carrier signal modulation, 63
ceiling function, bits per symbol, 172
certificate, license key, 71
circular clipping, 188, 192
clipping
  circular, 188, 192
  intermodulation distortion, 191
  peak-to-average power, 192
  power peaks, 189
  rectangular, 188, 192
  spectral regrowth, 191
coefficients, 113, 142
comments (instrument state register), 68
configuration of hardware
  Custom Arb, 126
  Real Time I/Q, 168
connectors
  digital bus, 30
  input
    10 MHz EFC, 32
    10 MHz IN, 32

connectors (*continued*)
  input
    AC power receptacle, 25
    DATA CLOCK, 18
    DATA INPUT, 17
    EXT 1 INPUT, 14
    EXT 2 INPUT, 14
    I/Q INPUTS, 17
    PULSE/TRIGGER GATE INPUT, 16
    SYMBOL SYNC INPUT, 18
    TRIGGER IN, 27, 28, 30, 31, 32
  interface
    AUXILIARY INTERFACE, 25
    GPIB, 25
    LAN, 26
    RS-232, 25
  output
    10 MHz OUT, 32
    LF OUTPUT, 14
    PULSE SYNC OUT, 15
    PULSE VIDEO OUT, 15
    RF OUTPUT, 15
    SOURCE SETTLED OUTPUT, 27
    SWEEP OUT, 26
    TRIGGER OUT, 26
continuous step sweep example, 40, 43
continuous wave output, 34
contrast hardkeys
  decrease, 17
  increase, 17
correction array (user flatness)
  configuration, 81
  load from step array, 81
  viewing, 82
  See also user flatness correction
correction, flatness. *See* user flatness correction
coupling
  factor, external detector, 75

D

DATA CLOCK connector, 18
data fields
  editing, 65
DATA INPUT connector, 17
data pattern creation, 134

Index

examples (*continued*)
  user flatness correction
    recalling data from memory, 83
    saving data to memory, 83
EXT 1 INPUT connector, 14
EXT 2 INPUT connector, 14
EXT annunciator, 20
EXT REF annunciator, 20
EXT1 annunciators
  HI, 20
  LO, 20
EXT2 annunciators
  HI, 20
  LO, 20
External DATA CLOCK
  setting to Normal or Symbol, 169

F

fail-safe recovery sequence, 241
failures. *See* troubleshooting
features
  PSG-A, 3, 4
  PSG-L, 2
files
  using, 67–70
  See also instrument state register
  See also memory catalog
filters
  Custom Arb, 110
  gaussian, loading default, 113, 142
  Real Time I/Q, 139
FIR, 66
FIR filter optimizing, 141
FIR filters, 110
firmware, upgrading, 243
flatness correction. *See* user flatness correction
FM
  annunciator, 20
  configuration example, 96
  deviation, 96
  hardkey, 13
  rate, 96
format, generating, 61
frequency
  display area, 19

frequency (*continued*)
  hardkey, 12
  LF output, 100
    start and stop, swept-sine, 101
  modulation. *See* FM
  offset, 35
  ramp sweep, 44
  reference, 35
  RF output, setting, 34
front panel
  diagram, 11
  display, 19
  knob, 12
FSK, 66
FSK modulation type creation, 161
FSK modulation type modified, 162
FSK selection, 124, 154
fundamental operation *See* basic operation

G

Gaussian BbT adjustment, 112, 141
gaussian filter, loading default, 113, 142
Gaussian selection, 111, 141
Goto Row softkey, 65
GPIB
  connector, 25
  listener mode, 84

H

hardkeys, 17
  ΦM, 13
  Amplitude, 12
  arrow, 16
  contrast
    decrease, 17
    increase, 17
  FM, 13
  Frequency, 12
  Help, 13
  Hold, 16
  Incr Set, 16
  Local, 17
  MENUS group, 13
  Mod On/Off, 14

Index

Index

Index

user flatness correction (*continued*)
  mm-wave source module, using
    recall correction data, 90
    save correction data, 90
    signal generator configuration, 87
  power meter
    configuration, 79
    models, 78
  signal generator configuration, 81
user-defined
  FIR filter, 115
  FIR filter creation, 144
user-defined modes
  recalling, 109
  selecting multicarrier, 107
  selecting single-carrier, 106

W

waveforms
  clipping, 188
  downloading, 207
  markers, 195
  multitone
    creating more than 64 tones, 214
    example, 215
    minimizing carrier feedthrough, 219, 221
    overview, 214
  player, overview, 182
  playing downloaded, 211
  segments, creating, 182
  segments, playing, 182
  segments, renaming, 182
  segments, storing and loading, 185
  sequence, building, 186
  triggers, 205
  two-tone
    example, 225
    minimizing carrier feedthrough, 227
    overview, 224
    signal alignment, 229
WFM1, 66