Measurement Guide and Programming Examples

N9073A-1FP W-CDMA Measurement Application
N9073A-2FP HSDPA/HSUPA Measurement Application

For use with the Agilent N9020A MXA and N9010A EXA Signal Analyzers

Agilent Technologies

Manufacturing Part Number: N9073-90006
Supersedes: N9073-90003

Printed in USA

August 2007

Legal Information

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

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

Where to Find the Latest Information

Documentation is updated periodically.

- For the latest information about Agilent Technologies Signal Analyzers, including software upgrades and application information, go to:

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

Technology Licenses

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

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as “Commercial computer software” as defined in DFAR 252.227-7014 (June 1995), or as a “commercial item” as defined in FAR 2.101(a) or as “Restricted computer software” as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies’ standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Contents

Contents

1 Making W-CDMA with HSDPA/HSUPA Measurements

This chapter begins with instructions common to all measurements, then details all the measurements available by pressing the **Meas** key when the **W-CDMA with HSDPA/HSUPA** mode is selected. For information specific to individual measurements, see the sections at the page numbers below.

Setting up and Making a Measurement

Making the Initial Signal Connection

CAUTION Before connecting a signal to the instrument, make sure the instrument can safely accept the signal level provided. The signal level limits are marked next to the connectors on the front panel.

See the Input Key menu for details on selecting input ports and the AMPTD Y Scale menu for details on setting internal attenuation to prevent overloading the instrument.

Using Instrument Mode and Measurement Presets

To set your current measurement mode to a known factory default state, press Mode Preset. This initializes the instrument by returning the mode setup and all of the measurement setups in the mode to the factory default parameters.

To preset the parameters that are specific to an active, selected measurement, press **Meas Setup**, **Meas Preset**. This returns all the measurement setup parameters to the factory defaults, but only for the currently selected measurement.

The 3 Steps to Set Up and Make Measurements

All measurements can be set up using the following three steps. The sequence starts at the Mode level, next is the Measurement level, then the result display may be adjusted.

1. **Select and Set Up the Mode**

   Press **Mode**. All licensed, installed modes available are shown. Press **W-CDMA with HSDPA/HSUPA**.

   Press **Mode Setup.** Make any required adjustment to the mode settings. These settings apply to all measurements in the mode.

2. **Select and Set Up the Measurement**

   Press **Meas**. Select the specific measurement to be performed (for example **ACP** or **Channel Power**). The measurement begins as soon as any required trigger conditions are met. The resulting data is shown on the display or is available for export.

   Press **Meas Setup.** Make any required adjustment to the selected measurement settings. The settings only apply to this measurement.

3. **Select and Set Up a View of the Results**

Press **View/Display**. Select a display format for the current measurement data. Depending on the mode and measurement selected, other graphical and tabular data presentations may be available. **X-Scale** and **Y-Scale** adjustments may also be made now.

NOTE A setting may be reset at any time, and will be in effect on the next measurement cycle or view.

Step	Primary Key	Setup Keys	Related Keys
1. Select and set up a mode.	**Mode**	**Mode Setup**, **FREQ Channel**	**System**
2. Select and set up a measurement.	**Meas**	**Meas Setup**	**Sweep/Control**, **Restart**, **Single**, **Cont**
3. Select and set up a view of the results.	**View/Display**	**SPAN X Scale**, **AMPTD Y Scale**	**Peak Search**, **Quick Save**, **Save**, **Recall**, **File**, **Print**

2 Channel Power Measurements

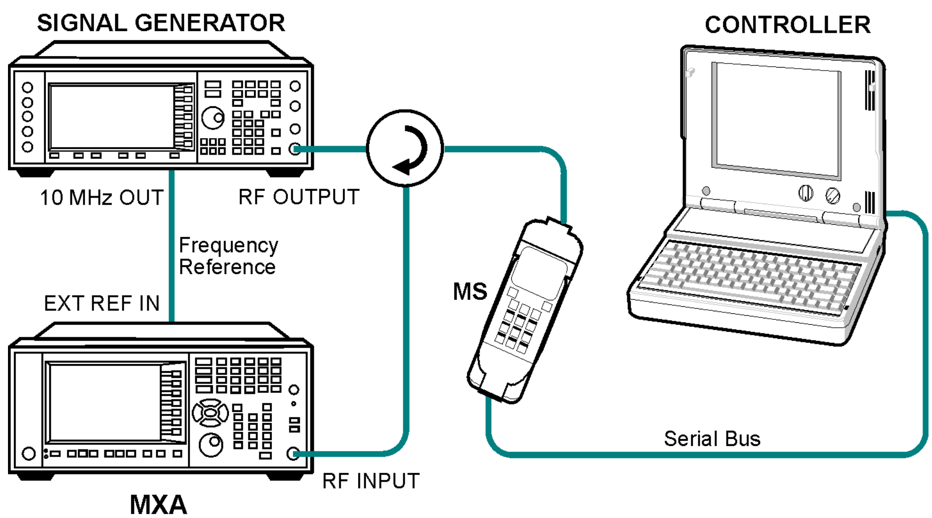
This chapter explains how to make a channel power measurement on a W-CDMA mobile station (MS). This test measures the total RF power present in the channel. The results are shown in a graph window and in a text window. If you install the optional HSDPA/HSUPA measurement application license, Code Domain and Modulation Accuracy can measure HSDPA/HSUPA signals as well.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 2-1 Channel Power Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal from the MS to the RF input port of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency:	1920 MHz (Channel Number: 5 × 1920 = 9600)
Output Power:	–20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the Mode.

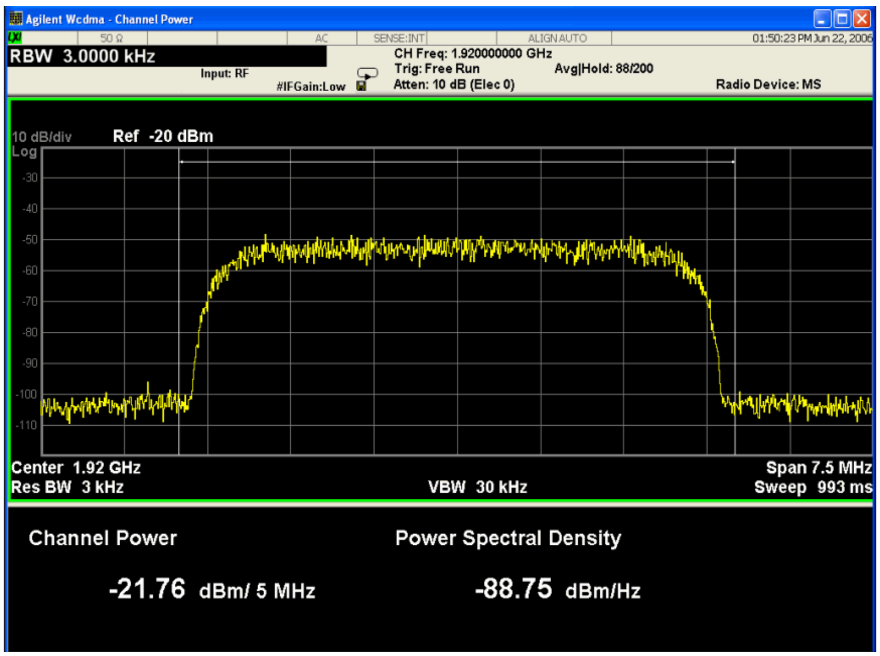
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Channel Power** to initiate the channel power measurement.

The Channel Power measurement result should look similar to Figure 2-2. The graph window and the text window show the absolute power and its mean power spectral density values over 5 MHz.

Figure 2-2 Channel Power Measurement Result



Step 6. Press **Meas Setup** to see the keys that are available to change measurement parameters from their default condition.

If you have a problem, and get an error message, see the "*Error Messages Guide*".

Channel Power Measurements

3 ACP Measurements

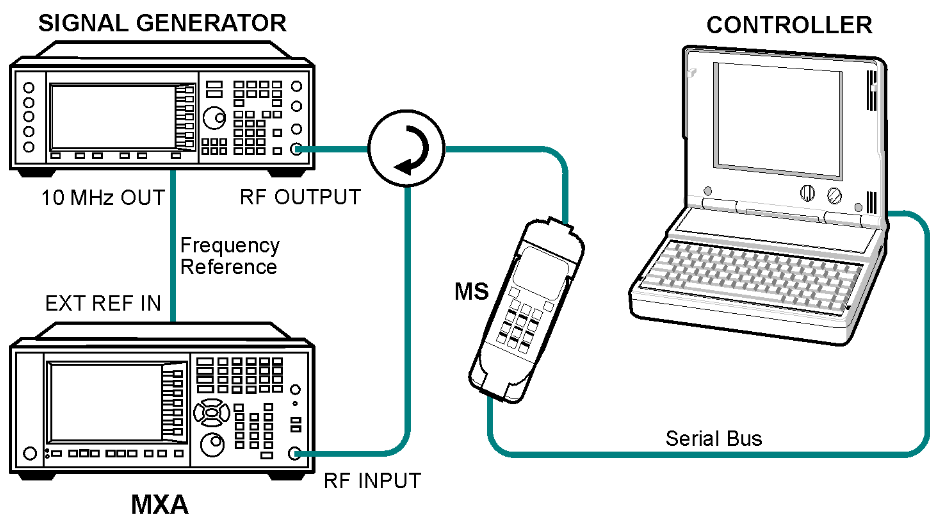
This chapter explains how to make the adjacent channel leakage power ratio (ACLR or ACPR) measurement on a W-CDMA mobile station (MS). ACPR is a measurement of the amount of interference, or power, in an adjacent frequency channel. The results are shown as a bar graph or as spectrum data, with measurement data at specified offsets.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 3-1 Adjacent Channel Power Ratio Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal from the MS to the RF input port of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1920 MHz (Channel Number: $5 \times 1920 = 9600$)

Physical Channels: DPCCH with 4 DPDCH

Scramble Code: 0

Output Power: –20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

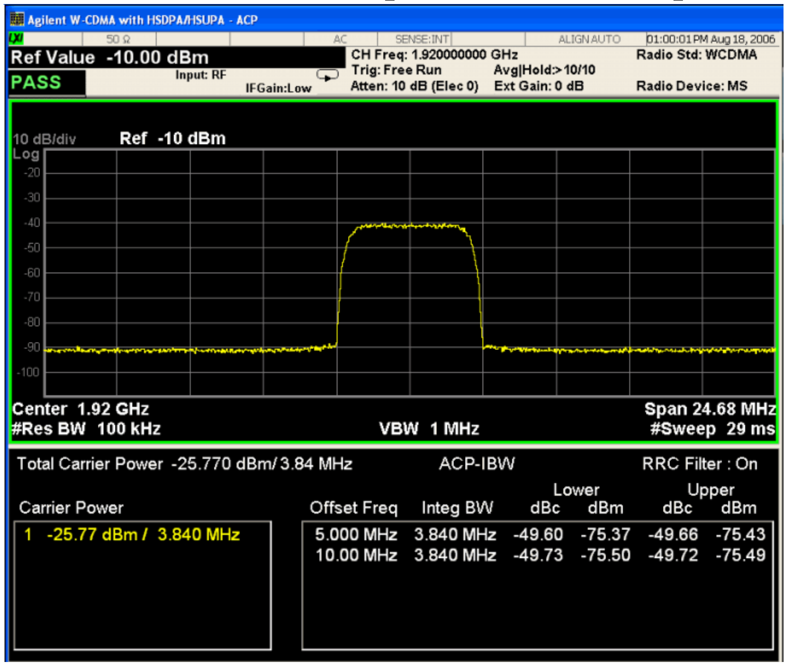
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **ACP** to initiate the adjacent channel leakage power ratio measurement.

Step 6. Press **View/Display**, and toggle the **Bar Graph** key to **Off** to see the spectrum trace graph.

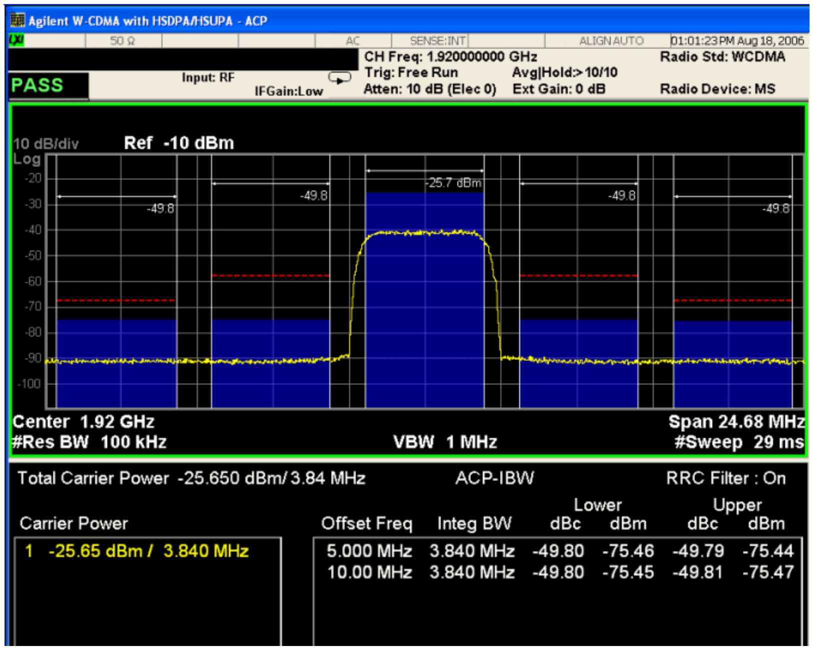
Figure 3-2 Measurement Result - Spectrum Trace Graph View



The spectrum graph measurement result should look similar to Figure 3-2. The graph (referenced to the total power) and a text window are displayed. The text window shows the absolute total power reference, while the lower and upper offset channel power levels are displayed in both absolute and relative readings.

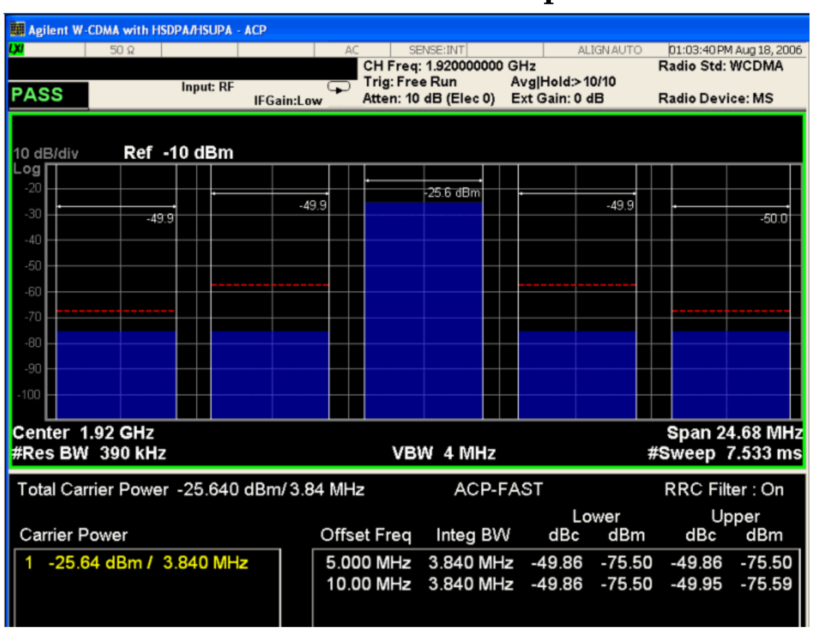
Step 7. Press **View/Display**, and toggle the **Bar Graph** key to **On** to see the bar graph with the spectrum trace graph overlay. The corresponding measured data is also shown in the text window. See Figure 3-3.

Figure 3-3 Measurement Result - Bar Graph View (Default)



Step 8. Press **Meas Setup**, **Meas Method**, and select **Fast**. The measurement result display is shown in Figure 3-4.

Figure 3-4 Fast Measurement Result - Bar Graph View



Step 9. Press **Meas Setup** to see the keys that are available to change the measurement parameters from the default condition.

If you have a problem, and get an error message, see the "*Error Messages Guide*".

4 Spectrum Emission Mask Measurements

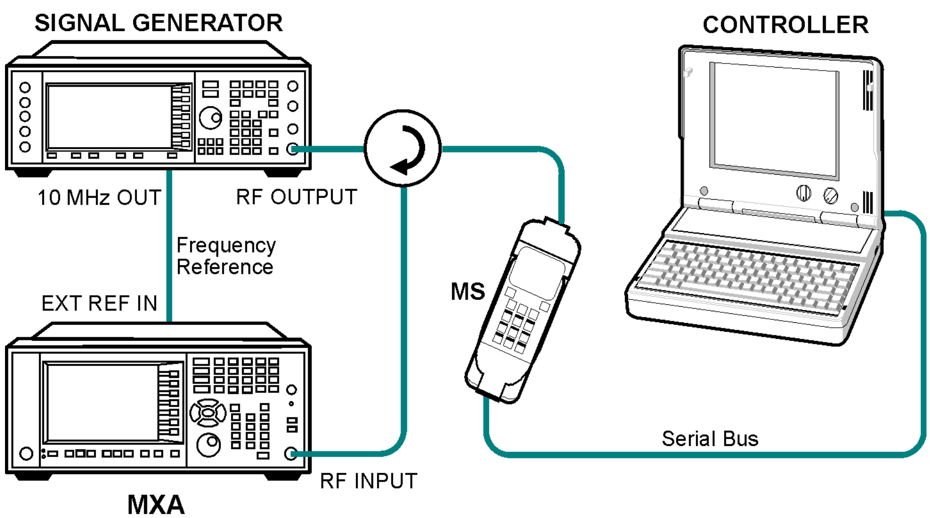
This chapter explains how to make the spectrum emission mask (SEM) measurement on a W-CDMA mobile station (MS). SEM compares the total power level within the defined carrier bandwidth and the given offset channels on both sides of the carrier frequency, to levels allowed by the standard. Results of the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

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 4-1 Spectrum Emission Mask Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal from the MS to the RF input port of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the base transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1920 MHz (Channel Number: 5 × 1,920 = 9,600)

Output Power: 0 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

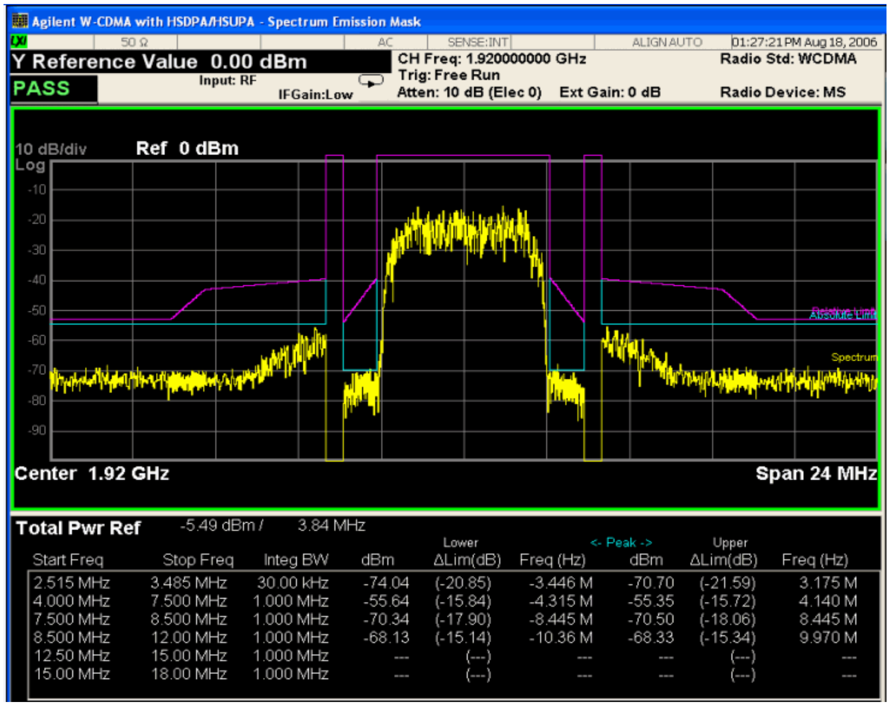
Step 2. Press **Mode Preset** to preset the mode.

Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Spectrum Emission Mask** to initiate the spectrum emission mask measurement.

Figure 4-2 Spectrum Emission Mask Measurement Result



The Spectrum Emission Mask measurement result should look similar to Figure 4-2. The text window shows the reference total power and the absolute peak power levels which correspond to the frequency bands on both sides of the reference channel.

If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

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

- Faulty DC power supply control of the transmitter power amplifier.

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

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

5 Spurious Emissions Measurement

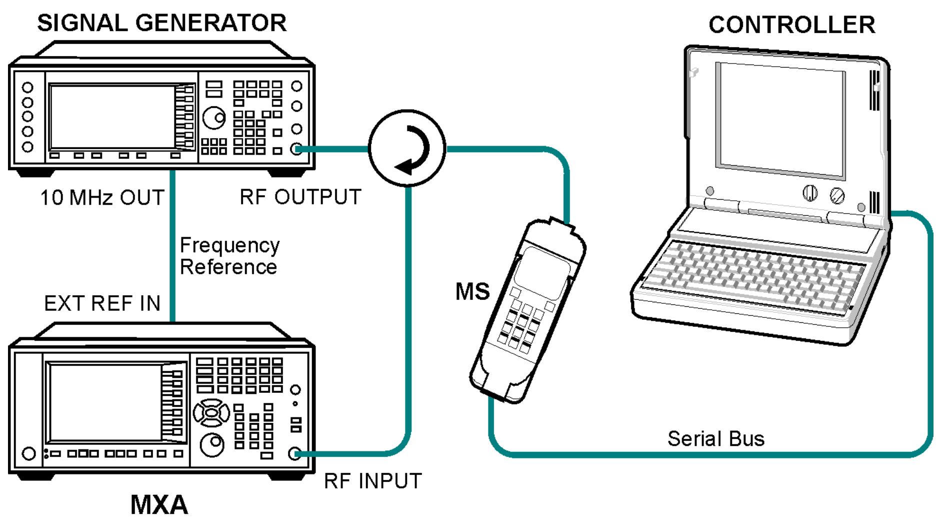
This section explains how to make the spurious emission measurement on a W-CDMA mobile station (MS). This measurement identifies and determines the power level of spurious emissions in certain frequency bands.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 5-1 Mobile Station Equipment Measurement System Setup



1. Using the appropriate cables, adapters, and circulator, connect the output signal from the MS to the RF input port of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting up the MS (Example)

From the system controller, perform all of the call acquisition functions required for the MS to transmit the RF power as required.

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

Step 3. Press **Input/Output**, **RF Input**, **RF Coupling** to toggle the RF Coupling to **DC**.

Step 4. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 5. Press **FREQ Channel**, enter a numerical frequency using the front-panel keypad, and select a units key, such as **MHz**.

Step 6. Press **Meas**, **Spurious Emission** to initiate the spurious emission measurement.

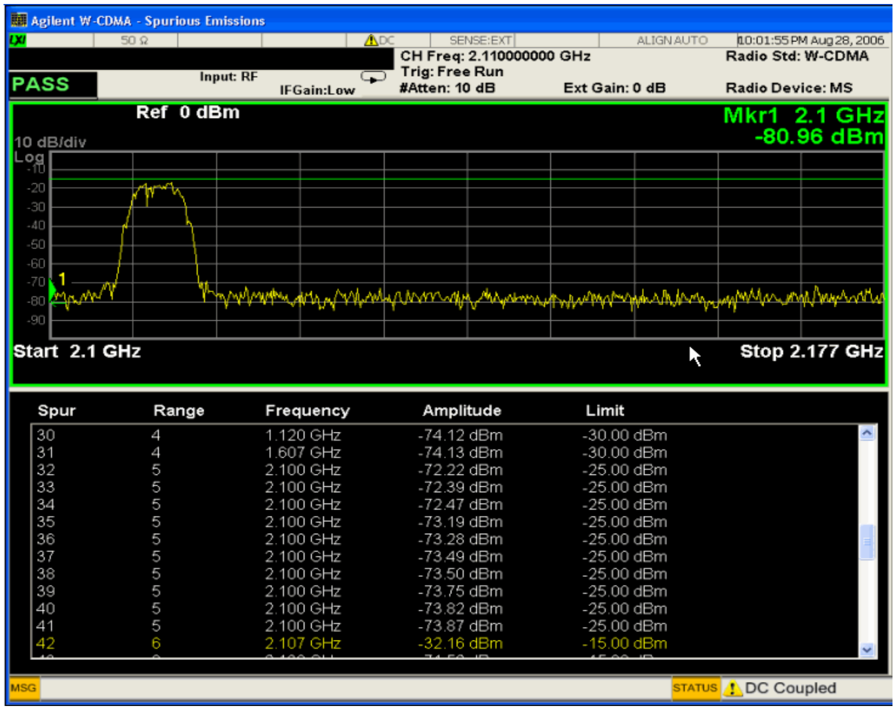
Depending on the current settings, the instrument will begin making the selected measurements. The resulting data is shown on the display or available for export.

If you want to change the measurement parameters from their default condition for a customized measurement, press **Meas Setup** to see the parameter keys that are available.

Measurement Results

The Spurious Emissions measurement results should look similar to Figure 5-2. The spectrum window and the text window show the spurs that are within the current value of the Marker Peak Excursion setting of the absolute limit. Any spur that has failed the absolute limit will have an 'F' beside it.

Figure 5-2 **Spurious Emissions Measurement**



If you have a problem, and get an error message, see the “*Error Messages Guide*”.

6 Occupied Bandwidth Measurements

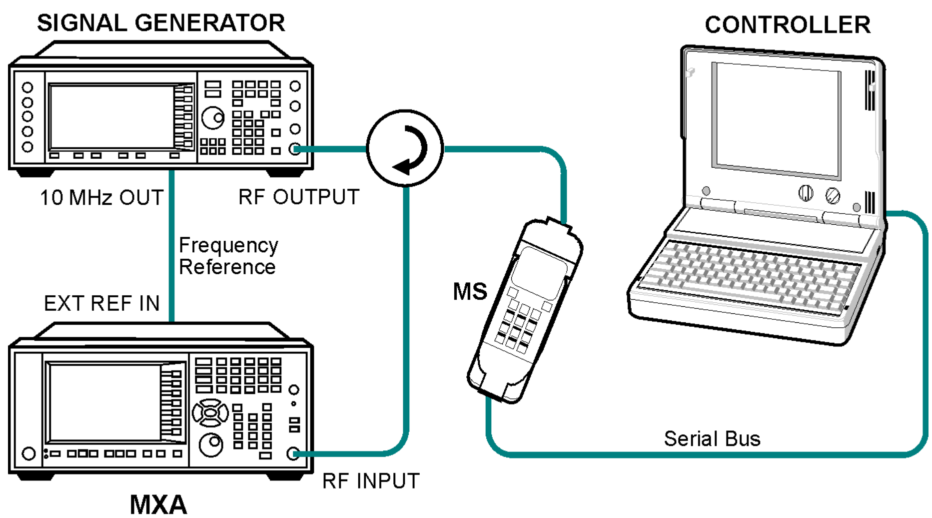
This chapter explains how to make the occupied bandwidth measurement on a W-CDMA mobile station (MS). The instrument measures power across the band, and then calculates its 99.0% power bandwidth.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 6-1 Occupied Bandwidth Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the base transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1920 MHz (Channel Number: $5 \times 1{,}920 = 9{,}600$)

Output Power: –20 dBm (or other power level for the MS)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

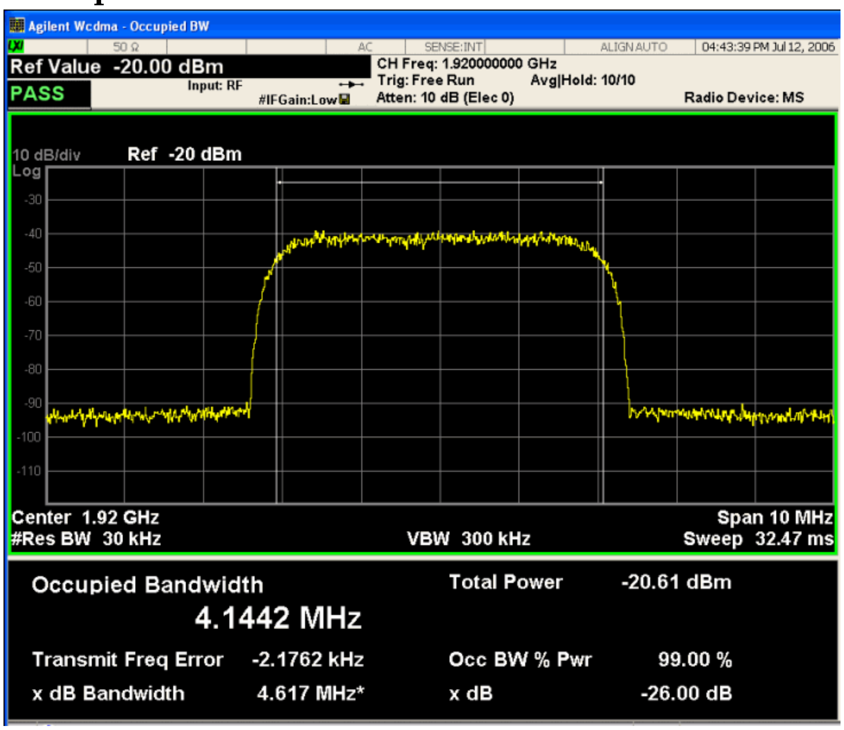
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Occupied BW** to initiate the occupied bandwidth measurement.

The Occupied BW measurement result should look similar to the Figure 6-2.

Figure 6-2 **Occupied Bandwidth Measurement Result**



If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

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

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

7 Power Statistics CCDF Measurements

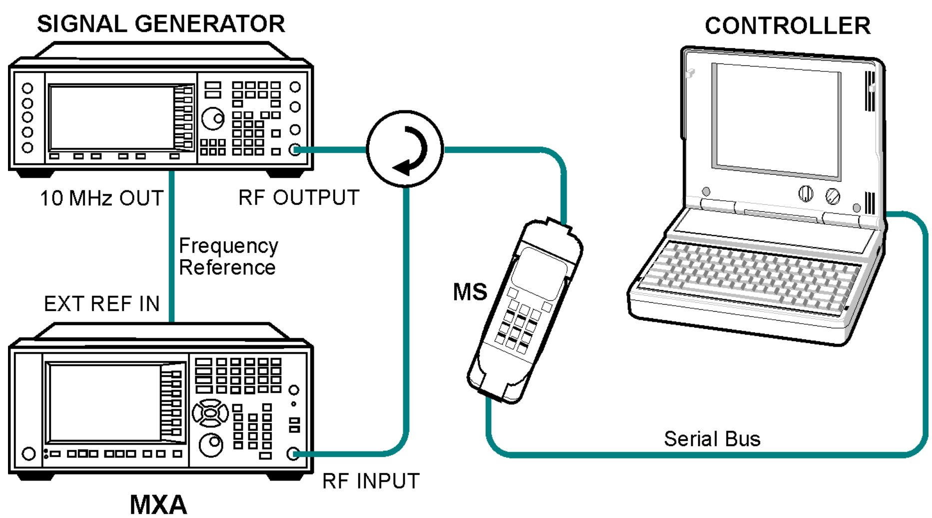
This section explains how to make the Power Statistics Complementary Cumulative Distribution Function (Power Stat CCDF) measurement on a W-CDMA mobile station (MS). Power Stat CCDF curves characterize the higher level power statistics of a digitally modulated signal.

Setting Up and Making a Measurement

Configuring the Measurement System

The mobile station (MS) under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 7-1 Power Statistics (CCDF) Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the base transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency:	1920 MHz (Channel Number: $5 \times 1{,}920 = 9{,}600$)
Physical Channels:	DPCCH with one or more DPDCH

Output Power: –20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

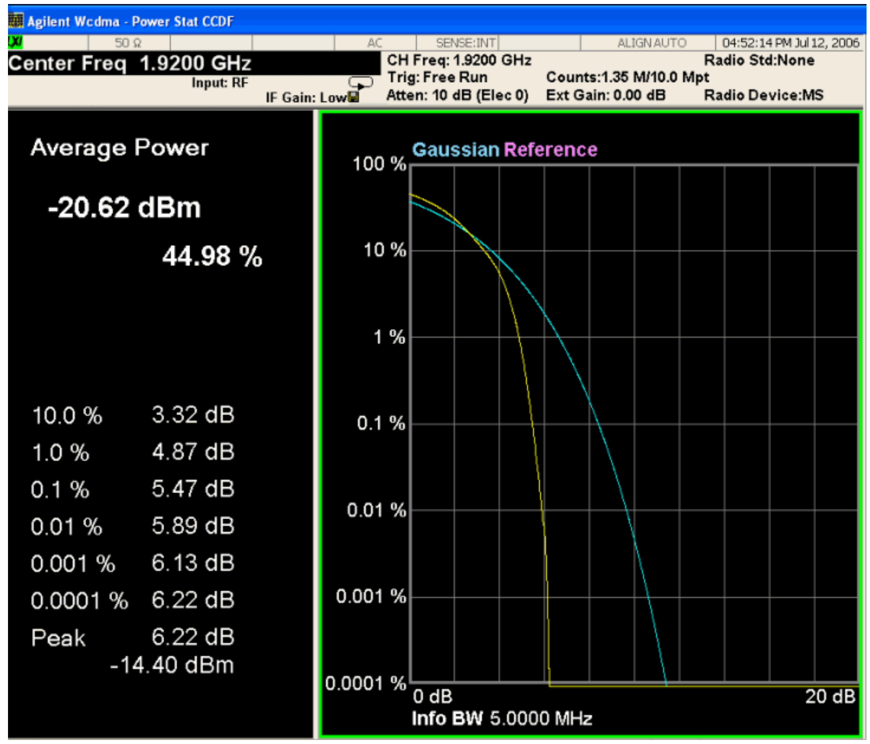
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Power Stat CCDF** to initiate the power statistics CCDF measurement.

The CCDF measurement result should look similar to Figure 7-2.

Figure 7-2 Power Statistics CCDF Result



If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

The power statistics CCDF measurement can contribute in setting the signal power specifications for design criteria for systems, amplifiers, and other components. For example, it can help determine the optimum operating point to adjust each code timing for appropriate peak or average power ratio, or both, throughout the wide channel bandwidth of the transmitter for a W-CDMA system.

8 Code Domain Measurements

This chapter explains how to make a code domain measurement on a W-CDMA mobile station (MS) and a base transceiver station (BTS). This is the measurement of power levels of the spread code channels across composite RF channels. The code power may be measured relative to the total power within the 3.840 MHz channel bandwidth, or absolutely, in units of power. Code Domain measurement examples using a W-CDMA uplink (UL) signal and a HSDPA downlink (DL) signal are shown in this section.

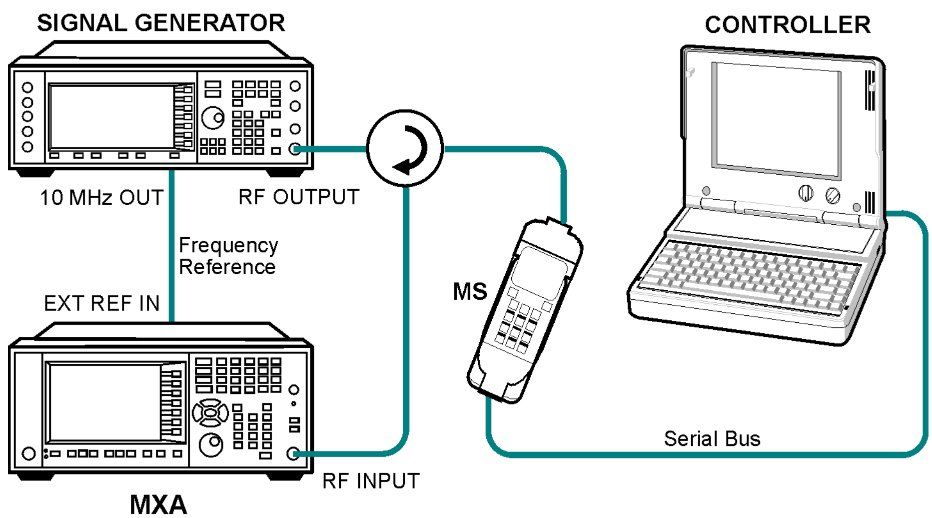
Code Domain Measurements

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 8-1 Code Domain Power Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the instrument.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the instrument.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

W-CDMA UL Measurement Example (Normal Mode)

Setting the MS (Example)

From the mobile station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1,920 MHz (Channel Number: 5 × 1,920 = 9,600)

Physical Channels: DPCCH with 4 DPDCH

Scramble Code: 0

Output Power: –20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

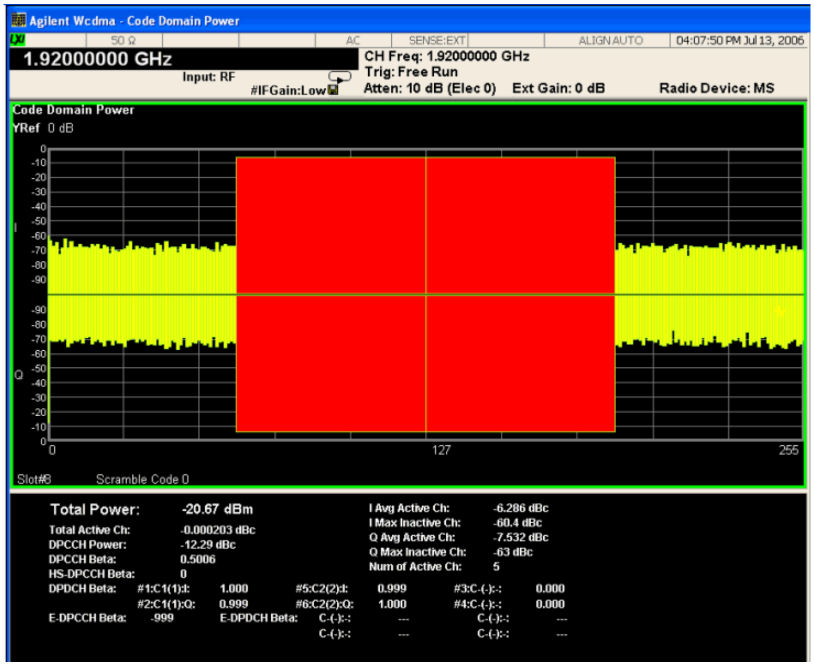
Step 2. Press **Mode Preset** to preset the mode.

Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Code Domain** to initiate the code domain measurement. The measurement result should look similar to Figure 8-2. The graph window is displayed with a text window below it. The text window shows the total power level along with the relative power levels of the various channels.

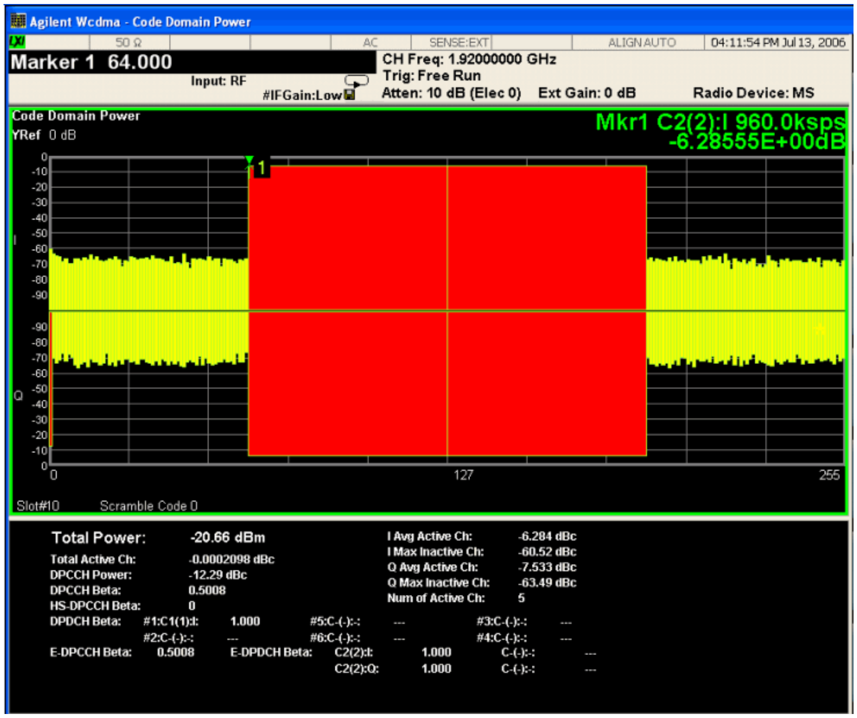
Figure 8-2 Code Domain Measurement Result - Power Graph & Metrics (Default) View



Step 6. Press **Peak Search** to put a marker on the highest power channel. See Figure 8-3. The I branch marker #1 measurement data shows C2(2), which indicates the code channel number 2 with SF 2^2 = 4. It also indicates the channel data rate at 960 ksps, and provides the power measurement of –6.28 dB in the channel relative to the total code power of the signal. The summary data shows active channels to be C1, C2, C5, C6. The Code Domain Power (CDP) and Code Domain Error (CDE) of the channel can be checked with the marker. For correct beta

calculation, **DPCH/E-DPCH Config** should be selected correctly under **Meas Setup**.

Figure 8-3 Code Domain Measurement Result - Power Graph & Metrics View - Uplink (MS) DPCCH, and 4 DPDCH w/ Peak Marker



Step 7. Press **Marker ->**, **Mkr-> Despread** to initiate the despreading and decoding of the marked channel to allow EVM and other error measurements to be conducted on the channel.

Step 8. Press **View/Display**, **Code Domain (Quad View)** to display a combination view of the code domain power, symbol power, and I/Q symbol polar vector graph windows, with a summary results window. See Figure 8-4.

Figure 8-4 Code Domain Measurement Result - Code Domain Quad View

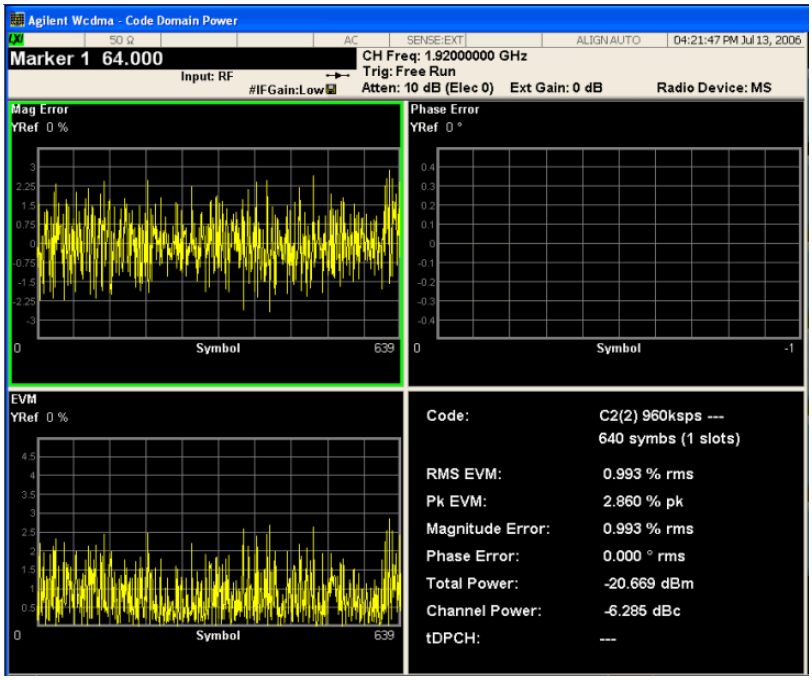
The original Code Domain Measurement is shown at the top left, while the Symbol Power measurement of the marked I-data channel is at the top right. The solid area below the first gradicule (blue on the instrument display) is the composite chip power versus time over the entire capture interval, while the (yellow) horizontal line is symbol power versus time for C2(2). The Capture Interval is 1 frame, but the measured interval is 1 slot.

The graph of the I/Q vector trajectory for C2(2) during the measurement interval is shown at lower left. As the constellation diagram shows, this example uses I-only data that is effectively BPSK modulation for channel C2(2), so the phase error must be zero.

The summary data at the lower right indicates peak and RMS EVM, magnitude and phase errors, powers of the signal and the channel.

Step 9. Press **View/Display**, **I/Q Error (Quad View)** to display a combination view of the magnitude error vs. symbol, phase error vs. symbol, and EVM vs. symbol graph windows, with the modulation summary results window. See Figure 8-5.

Figure 8-5 Code Domain Measurement Result - I/Q Error Quad View

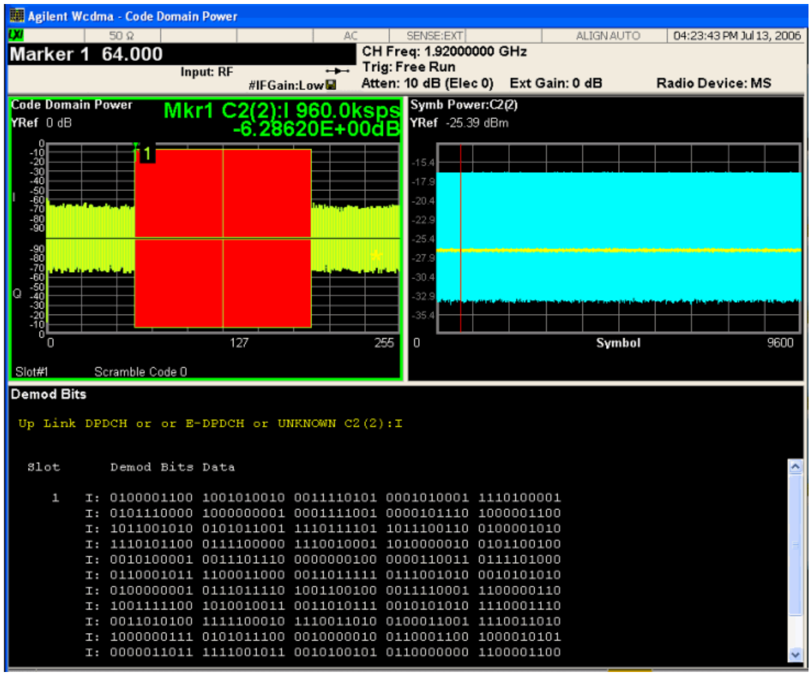


The results screen shows the data for the same code domain channel C2(2) that was selected for despreading by the marker in the Code Domain Quad View in the previous step.

Again, this example uses I-only data that is effectively BPSK modulation for channel C2(2), the phase error must therefore be zero.

Step 10. Press **View/Display, Demod Bits** to display a combination view of the code domain power, symbol power graph windows, and the I/Q demodulated bit stream data for the symbol power slots selected by the measurement interval and measurement offset parameters.

Figure 8-6 Code Domain Measurement Result - Demod Bits View



The Demod Bits View displays the same Code Domain Power and Symbol Power windows as the Code Domain (Quad View) shown in Figure 8-4 on page 37.

The demodulated bit stream displayed is the data contained in the Measurement Interval, slot #1. In the Symbol Power graph, this is the data between the red vertical lines; 1 slot, with no offset, so it is the first slot of the capture interval of 1 frame.

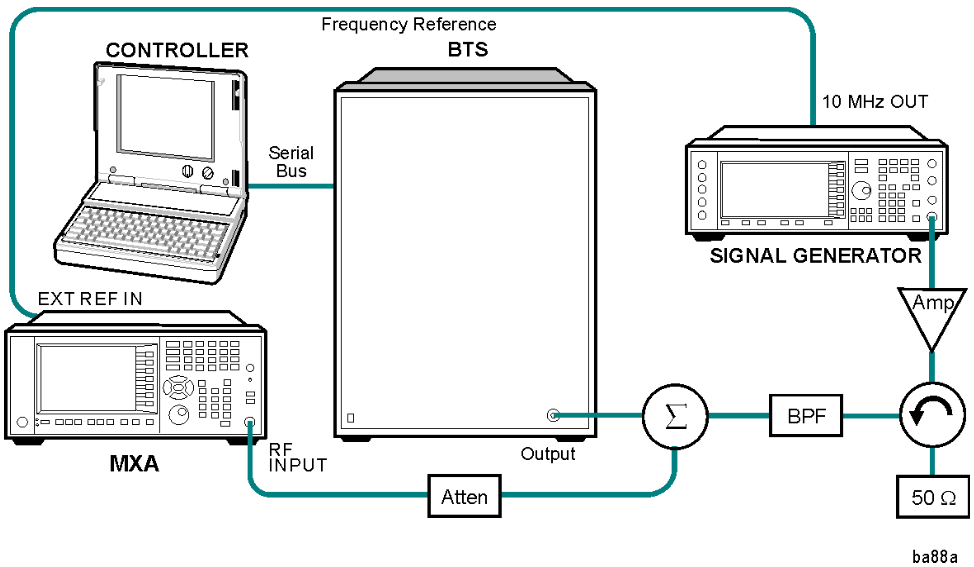
If you have a problem, and get an error message, see the "*Error Messages Guide*".

HSDPA DL Measurement Example (Test Model 5)

Configuring the Measurement System

Use the system controller to remotely control the base transceiver station (BTS) under test to transmit the RF power. The W-CDMA modulated interference signal is injected to the antenna output port of the BTS through an attenuator and circulator. The transmitting signal from the BTS is connected to the RF input port of the instrument from the circulator port. Connect the equipment as shown.

Figure 8-7 Intermodulation Product Measurement System



1. Using the appropriate amplifier, circulator, bandpass filter, combiner, cables, and adapters, connect the unmodulated carrier signal from the signal generator to the output connector of the BTS.
2. Connect the circulator output signal to the RF input port of the analyzer through the attenuator.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the BTS through the serial bus cable.

Setting the BTS (Example)

From the BTS simulator or the system controller, or both, perform all of the call acquisition functions required for the BTS to transmit the RF power as follows:

Frequency: 1000 MHz

Physical Channels: Test Model 5 with 8 HS-PDSCH

Scramble Code:	0
Output Power:	–10 dBm

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA w/HSDPA/HSUPA measurements.

Step 2. Press **Mode Preset** to preset the mode.

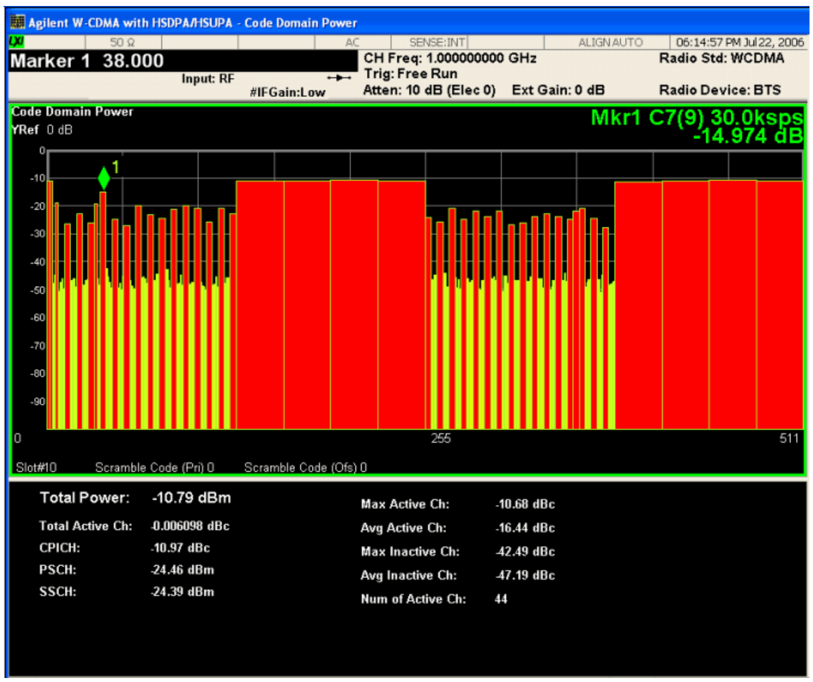
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **BTS**.

Step 4. Press **FREQ Channel**, **1000**, **MHz** to set the center frequency to 1.000 GHz.

Step 5. Press **Meas**, **Code Domain** to initiate the code domain measurement.

Step 6. Press **Meas Setup**, **Symbol Boundary**, **Predefined Test Models**, **Test Model 5**, **Test Model 5 w/ 8 HS-PDSCH w/30 DPCH**.

Figure 8-8 Code Domain Measurement Result - Power Graph & Metrics (Default) View - Downlink (BTS) Test Model 5



The Code Domain Power measurement result should look similar to Figure 8-8. The graph window is displayed with a text window below it. The text window shows the total power level along with the relative power levels of the various channels.

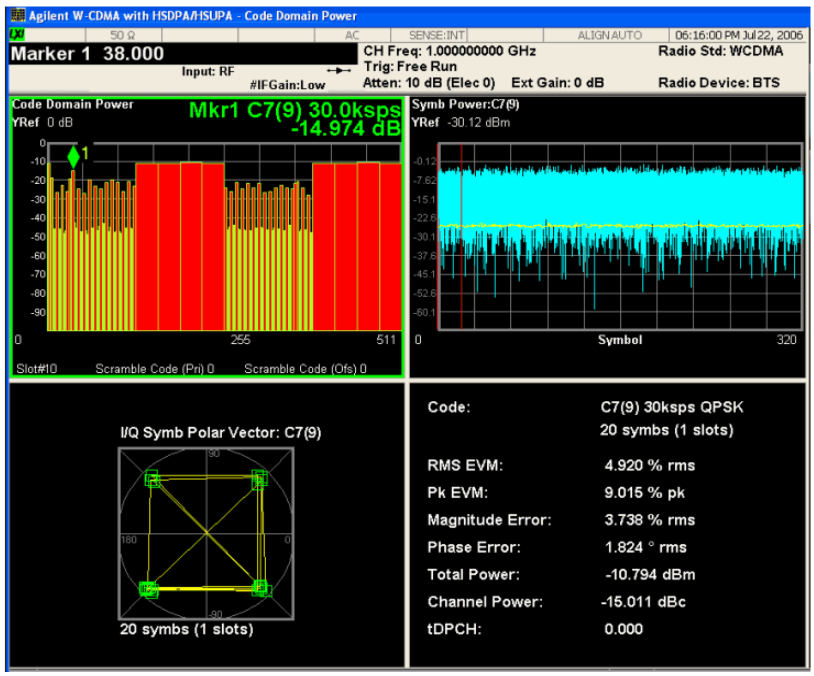
Now to examine a single HSDPA code channel in the code domain more closely:

Step 7. Press **Marker**, and enter **38** using the front panel keypad, **Enter**.

Step 8. Press **Marker**, **Mkr->**, **Mkr-> Despread** to initiate the despreading and decoding of the marked channel to allow EVM and other error measurements to be conducted on the channel.

Step 9. Press **View/Display**, **Code Domain (Quad View)** to display the combination view of the code domain power, symbol power, and I/Q symbol polar vector graph windows, and summary results window. See Figure 8-9.

Figure 8-9 Code Domain Measurement Result - Code Domain Quad View - HSDPA DL Test Model 5



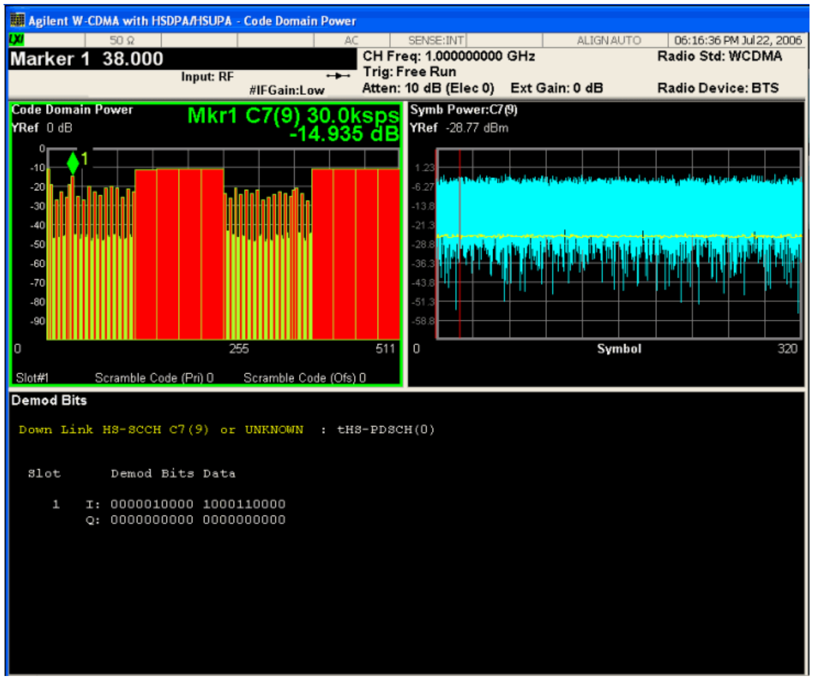
The original Code Domain Measurement with the marker at code channel 38 is shown at the top left, while the Symbol Power measurement of the marked channel is at the top right. The solid area below the first gradicule (blue on the instrument display) is the composite chip power over the entire capture interval, while the (yellow) horizontal line is Symbol power for C7(9).

The vertical red line in the graph indicates the measurement interval, with the default measurement offset of 0 slots. The graph of the I/Q vector trajectory for C7(9) during the measurement interval is shown at lower left. The summary data at lower right indicates peak and RMS EVM, magnitude and phase errors, powers of signal and channel.

TIP If your EVM or Phase Error results are high, and you have many code channels in your signal, try using the Multi Channel Estimator to improve your measurement result. Press **Meas Setup**, **Advanced**, and toggle the **Multi Channel Estimator** key to **ON**.

Step 10. Press **View/Display**, **Demod Bits** to display the combination view of the code domain power, symbol power graph windows, and the I/Q demodulated bit stream data for the symbol power slots selected by the measurement interval and measurement offset parameters.

Figure 8-10 Code Domain Measurement Result - Demod Bits View - HSDPA DL Test Model 5



The Demod Bits View displays the same Code Domain Power and Symbol Power windows as the Code Domain (Quad View) shown in Figure 8-9 on page 42.

The demodulated bit stream displayed is the data contained in the Measurement Interval (1 slot, with no offset, so it is the first slot) of the Capture Interval of 2 frames.

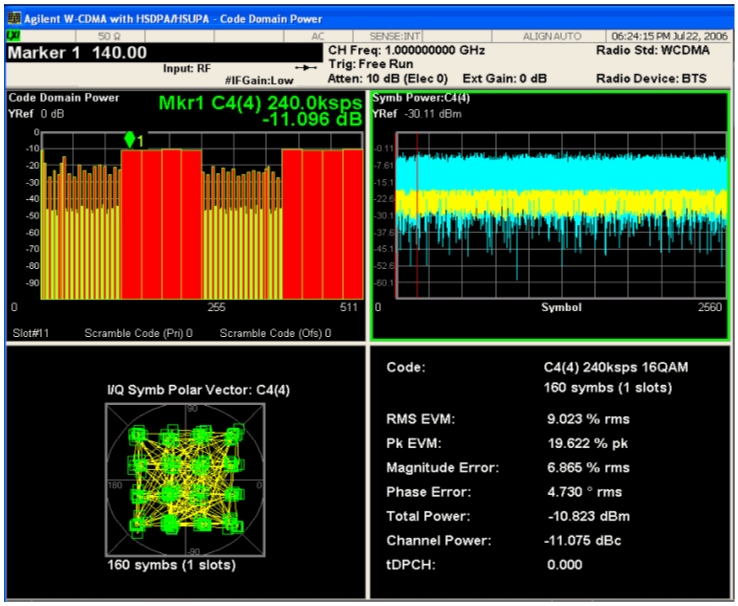
Step 11. Press **View/Display**, **Power Graph & Metrics** to display the primary code domain power view and summary results window for another HSDPA code channel, HS-PDCSH.

Step 12. Press **Marker**, and enter **140** using the front panel keypad, **Enter**.

Step 13. Press **Mkr->**, **Mkr-> Despread** to initiate the despreading and decoding of the marked channel to allow EVM and other error measurements to be conducted on the channel. It may be necessary to press **Restart** if the measurement setting is on **Single**.

Step 14. Press **View/Display**, **Code Domain (Quad View)** to display the combination view of the code domain power, symbol power, and I/Q symbol polar vector graph windows, and summary results window.

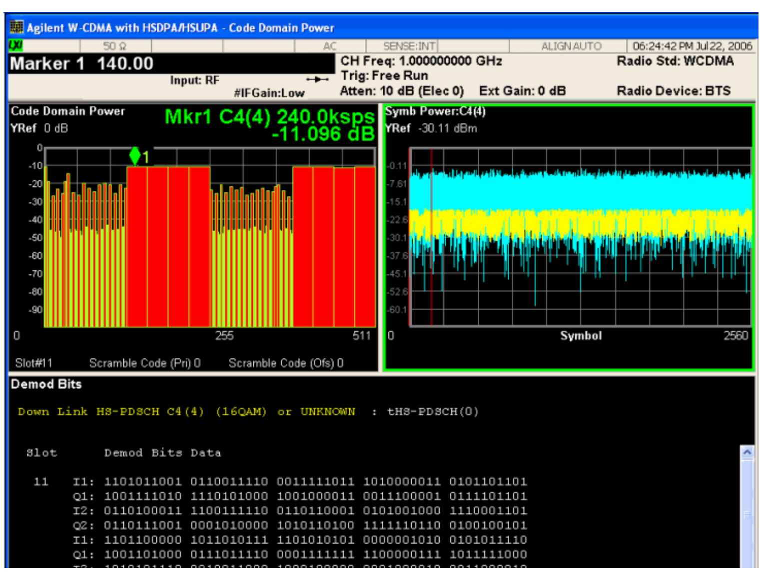
Figure 8-11 Code Domain Measurement Result - Code Domain Quad View - HSDPA DL Test Model 5



This code channel C4(4) is the HS-PDSCH, unique to HSDPA, and present in Test Model 5. The difference in symbol power can be clearly seen. The 16QAM modulation is also displayed, instead of the normal QPSK for W-CDMA DPCH channels.

Step 15. Press **View/Display**, **Demod Bits** again to display the I/Q demodulated bit stream data for the symbol power slots selected by the measurement interval and measurement offset parameters.

Figure 8-12 Code Domain Measurement Result - Demod Bits View (Binary)



The demodulated bits for slot 11 are shown in Binary format. You can also view the bit stream in Hexadecimal format by doing the following:

Using the Next Window key, move the view to the Demod Bits screen. Press **View/Display**, **Demod Bits**, **Demod Bits Format** to toggle to Hex format

If you have a problem, and get an error message, see the “*Error Messages Guide*”.

Troubleshooting Hints

Uncorrelated interference may cause CW interference like local oscillator feed through or spurs. Another cause of uncorrelated noise can be I/Q modulation impairments. Correlated impairments can be due to the phase noise on the local oscillator in the upconverter or I/Q modulator of the unit under test (UUT). These will be analyzed by the code domain measurements along with the QPSK EVM measurements and others.

Poor phase error indicates a problem at the I/Q baseband generator, filter, or modulator in the transmitter circuitry of the UUT, or both. The output amplifier in the transmitter can also create distortion that causes unacceptably high phase error. In a real system, poor phase error will reduce the ability of a receiver to correctly demodulate the received signal, especially in marginal signal conditions.

9 Modulation Accuracy (Composite EVM) Measurements

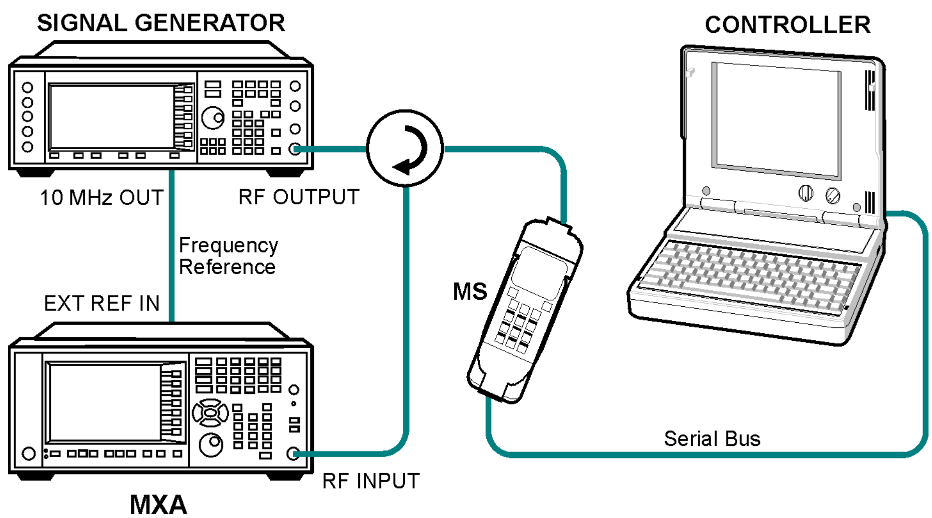
This section explains how to make the modulation accuracy (composite EVM) measurement on a W-CDMA mobile station (MS). Modulation accuracy is the ratio of the correlated power in a multi-coded channel to the total signal power.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 9-1 Modulation Accuracy Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS (Example)

From the base transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1920 MHz (Channel Number: $5 \times 1{,}920 = 9{,}600$)

Physical Channels: A coded signal with the DPCCH and at least one DPDCH is required to make a composite EVM measurement on a W-CDMA UL signal. (A

W-CDMA DL signal must contain either the SCH or the CPICH.)

Scramble Code: 0

Output Power: –20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Preset** to preset the mode.

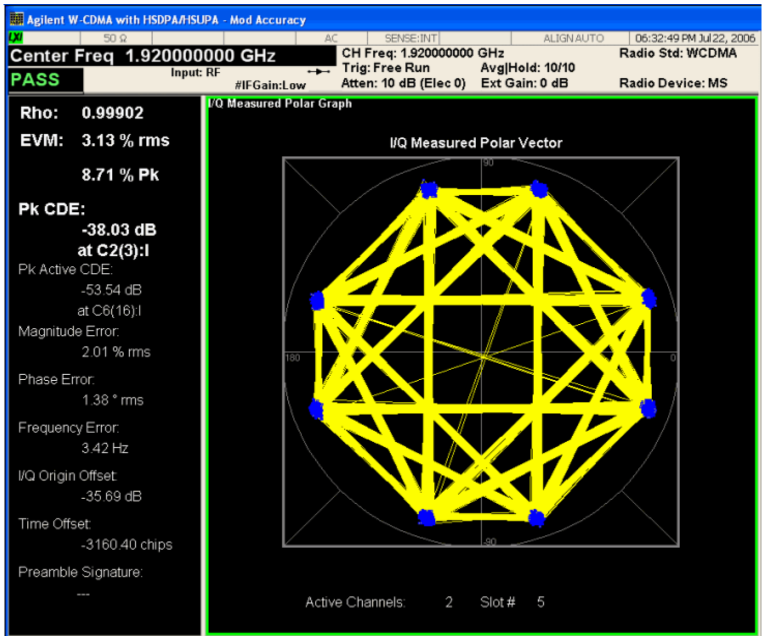
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **Mod Accuracy (Composite EVM)** to initiate the modulation accuracy (composite EVM) measurement.

The Mod Accuracy I/Q Polar Vector Constellation measurement result should look similar to Figure 9-2.

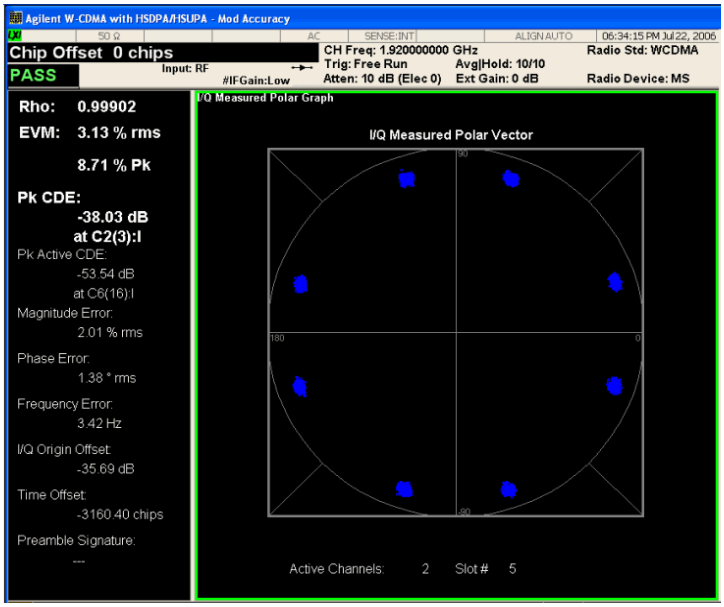
Figure 9-2 Modulation Accuracy Measurement Result - I/Q Measured Polar Graph (Default) View



The modulation constellation is shown, along with summary data for Rho, EVM, Peak Code Domain Error, and phase and magnitude errors.

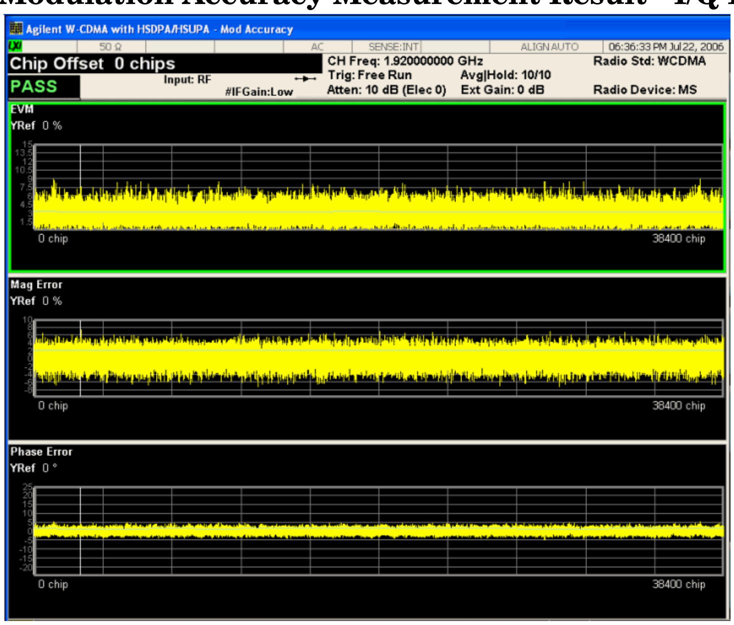
Step 6. Press **View/Display**, **I/Q Measured Polar Graph, I/Q PolarVec/Constln, Constellation** to display a view of the I/Q measured polar constellation graph window and the modulation summary result window.

Figure 9-3 Modulation Accuracy Measurement Result - Polar Constellation View



Step 7. Press **View/Display**, **I/Q Error** to display a combination view of the magnitude error, phase error, and EVM graph windows.

Figure 9-4 Modulation Accuracy Measurement Result - I/Q Error View



If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

A poor phase error often indicates a problem with the I/Q baseband generator, filters, or modulator, or all three, in the transmitter circuitry of the unit under test (UUT). The output amplifier in the transmitter can also create distortion that causes unacceptably high phase error. In a real system, a poor phase error will reduce the ability of a receiver to correctly demodulate the received signal, especially in marginal signal conditions.

If the error "`Can not correlate to input signal`" is shown, it means that your measurement has failed to find any active channels due to the lack of correlation with the input signal. The input signal level or scramble code, or both, may need to be adjusted to obtain correlation.

10 Power Control Measurements

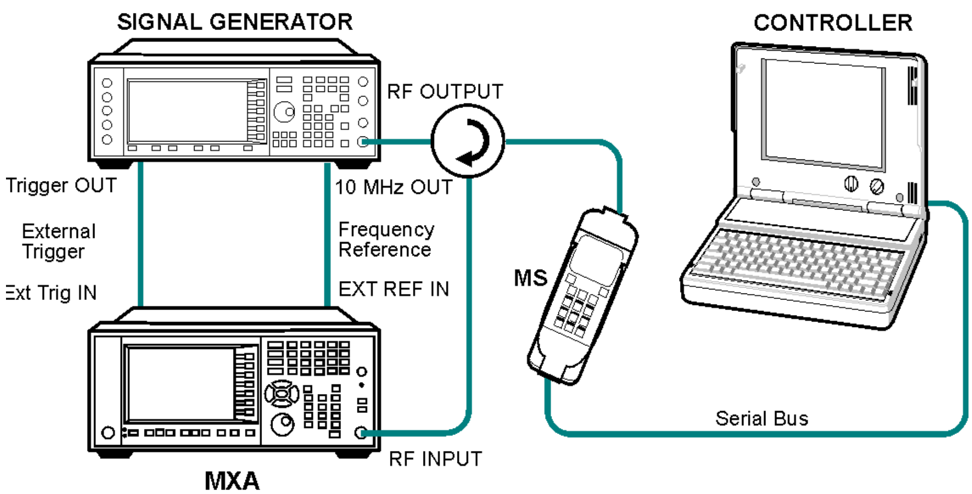
This chapter explains how to make power control measurements on W-CDMA mobile stations (MS). Power control measurements characterize the ability of a mobile station to vary the power levels of a digitally modulated signal, as directed by the base station. There are three selections of measurement type; Slot Power to monitor the power steps, PRACH Power to verify the PRACH preambles and PRACH message power levels, and Slot Phase for user equipment (UE) phase discontinuity.

Setting Up and Making a Measurement

Configuring the Measurement System

The MS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 10-1 Power Control Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect a trigger signal from the signal generator or system controller and the EXT TRIG IN port of the analyzer.
5. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS

From the transceiver station simulator or the system controller, or both, perform all of the call acquisition and power control functions required for the MS to transmit the RF power as follows:

Frequency: 1000 MHz

Physical Channels: DPCCH with one or more DPDCH

NOTE This example shows a signal with 4 dB power steps across the frame.

Output Power: –15 dBm (at analyzer input) peak

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

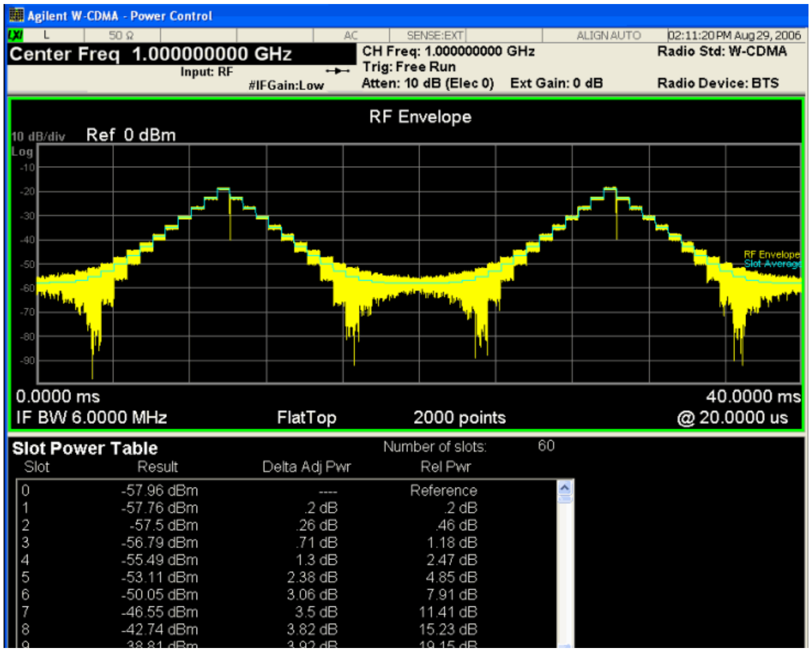
Step 4. Press **Trigger** to select the **External 1** or **External 2** trigger supplied.

Step 5. Press **FREQ Channel**, **1000**, **MHz** to set the center frequency to 1.000 GHz.

Step 6. Press **Meas**, **Power Control** to initiate the default power control measurement

The default power control measurement result should look similar to Figure 10-2.

Figure 10-2 Power Control Measurement Result - Slot Power Graph Metrics View



Step 7. To make measurements repeatedly, press the **Cont** front panel key.

If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

The power control measurement, along with the power versus time measurement and spectrum measurement, can reveal the effects of degraded or defective parts in the transmitter section of the unit under test (UUT). The following are areas of concern which can contribute to performance degradation:

- DC power supply control of the transmitter power amplifier, RF power control of the pre-power amplifier stage, or I/Q control of the baseband stage, or all.
- Gain and output power levels of the power amplifier, caused by degraded gain control or increased distortion, or both.
- Amplifier linearity.

11 QPSK EVM Measurements

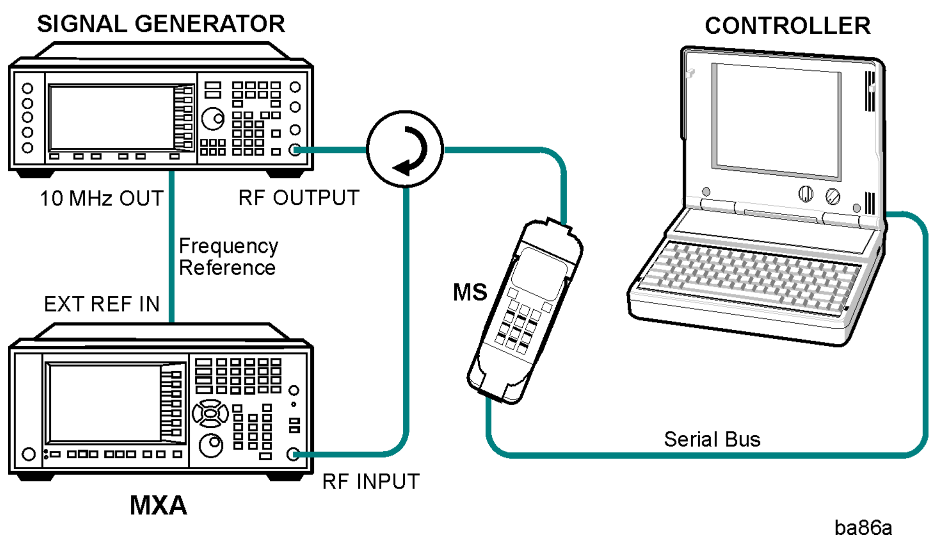
This chapter explains how to make the QPSK error vector magnitude (EVM) measurement on a W-CDMA mobile station (MS). QPSK EVM is a measure of the phase and amplitude modulation quality that relates the performance of the actual signal compared to an ideal signal as a percentage, calculated over the course of the ideal constellation.

Setting Up and Making a Measurement

Configuring the Measurement System

The mobile station (MS) under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 11-1 QPSK EVM Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the base transceiver station simulator or signal generator to the MS through the circulator to initiate a link constructed with the sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

Setting the MS

From the base transceiver station simulator or the system controller, or both, perform all of the call acquisition functions required for the MS to transmit the RF power as follows:

Frequency: 1920 MHz (Channel Number: $5 \times 1{,}920 = 9{,}600$)

Physical Channels: DPCCH only

Scramble Code:	0
Output Power:	–20 dBm (at analyzer input)

Measurement Procedure

Step 1. Press **Mode**, **W-CDMA with HSDPA/HSUPA** to enable the W-CDMA measurements.

Step 2. Press **Mode Preset** to preset the mode.

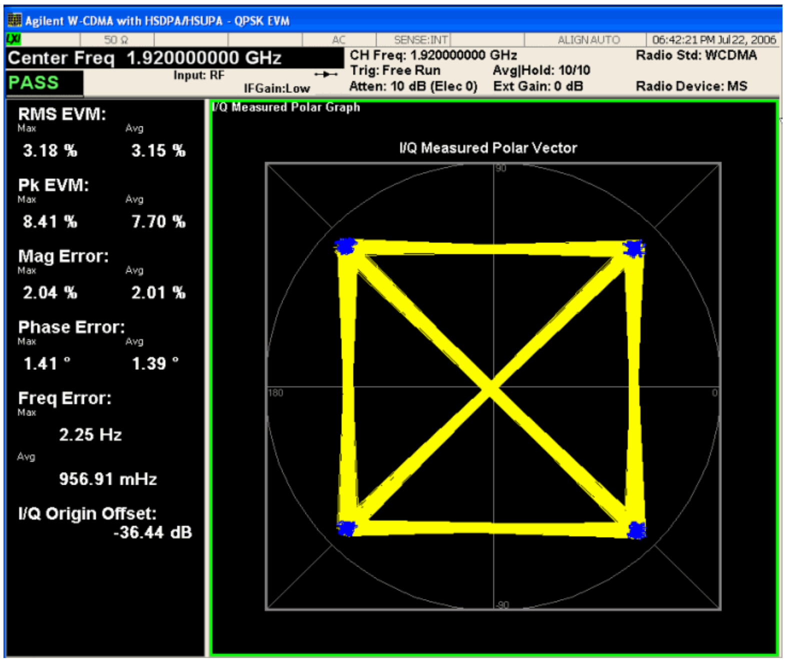
Step 3. Press **Mode Setup**, **Radio**, **Device** to toggle the device to **MS**.

Step 4. Press **FREQ Channel**, **1920**, **MHz** to set the center frequency to 1.920 GHz.

Step 5. Press **Meas**, **QPSK EVM** to initiate the QPSK EVM measurement.

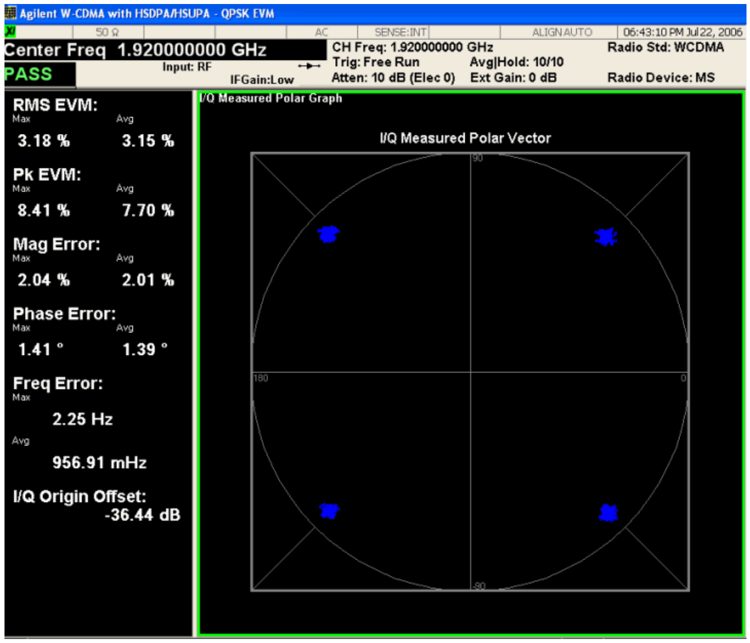
The QPSK EVM I/Q Measured Polar Vector measurement result should look similar to Figure 11-2. The measurement values for modulation accuracy are shown in the summary result window.

Figure 11-2 QPSK EVM Result - Polar Vector/Constellation (Default) View



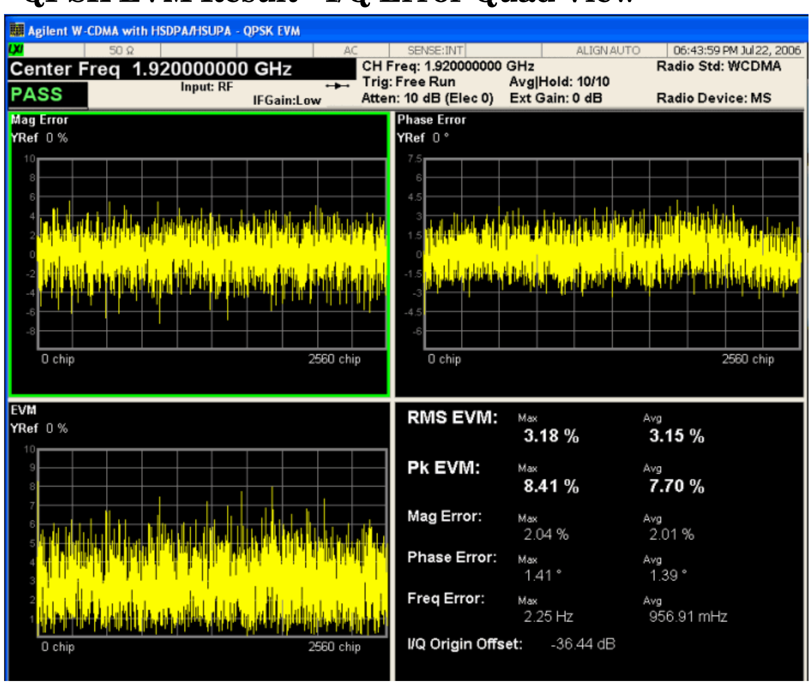
Step 6. Press **View/Display**, **I/Q Measured Polar Graph**, **I/Q Polar Vec/Constln**, **Constellation** to display a view of the I/Q measured polar constellation graph window and the modulation summary result window.

Figure 11-3 QPSK EVM Result - Polar Constellation View



Step 7. Press **View/Display**, **I/Q Error** to display a combination view of the magnitude error, phase error, EVM graph windows, and the modulation summary result window.

Figure 11-4 QPSK EVM Result - I/Q Error Quad View



If you have a problem, and get an error message, see the "*Error Messages Guide*".

Troubleshooting Hints

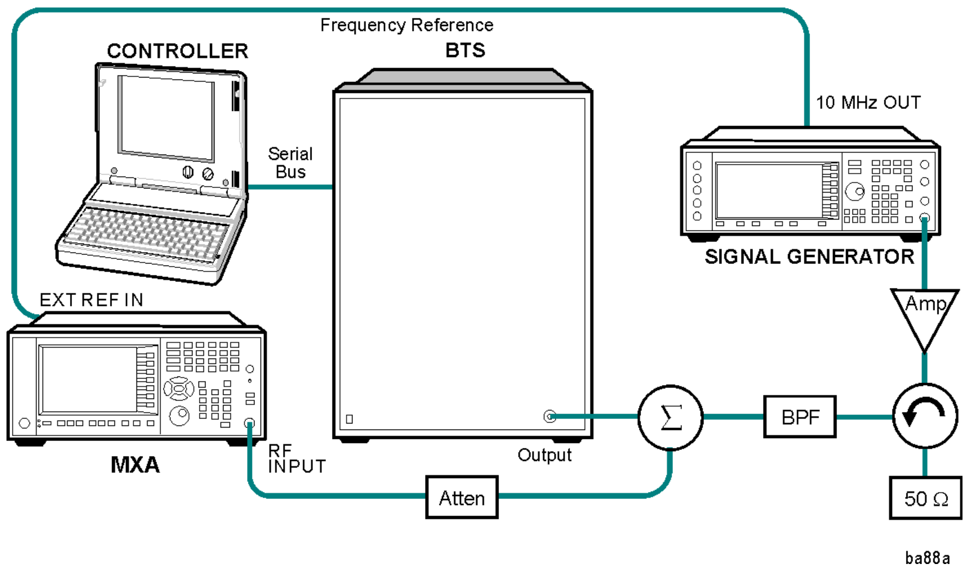
A poor phase error indicates a problem with the I/Q baseband generator, filters, or modulator, or all, in the transmitter circuitry of the unit under test (UUT). The output amplifier in the transmitter can also create distortion that causes unacceptably high phase error. In a real system, a poor phase error will reduce the ability of a receiver to correctly demodulate the received signal, especially in marginal signal conditions.

12 Monitor Spectrum Measurements

This chapter explains how to make a Monitor Spectrum measurement on a Mobile Station (MS). Monitor Spectrum measurements show a spectrum domain display of the signal.

This example shows a MS under test set up to transmit RF power, and controlled remotely by a system controller. The transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown.

Figure 12-1 Monitor Spectrum Measurement



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the MS to the RF input of the analyzer.
2. Connect the MS simulator or signal generator to the MS through a circulator to initiate a link constructed with sync and pilot channels, if required.
3. For best frequency accuracy, connect a BNC cable between the 10 MHz REF IN port of the signal generator (if available) and the 10 MHz EXT REF OUT port of the analyzer.
4. Connect the system controller to the MS through the serial bus cable to control the MS operation.

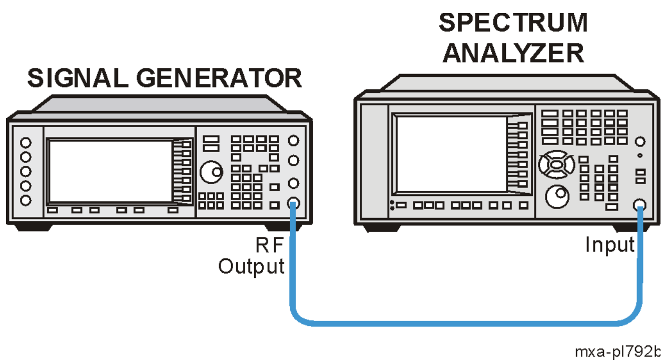
Measurement Procedure

Step 1. Setup the signal sources as follows:

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

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

Figure 12-2 Setup for Obtaining Two Signals



Step 3. Enable the W-CDMA Mode:

Press **Mode**, **W-CDMA with HSDPA/HSUPA**.

Step 4. Preset the analyzer mode:

Press **Mode Preset**.

Step 5. Set the measurement center frequency:

Press **FREQ Channel**, enter a numerical frequency using the front-panel keypad, and select a units key, such as **MHz**.

Step 6. Set the measurement span frequency:

Press **SPAN X Scale**, enter a numerical span using the front-panel keypad, and select a units key, such as **MHz**.

Step 7. Initiate the measurement:

Press **Meas**, **Monitor Spectrum**.

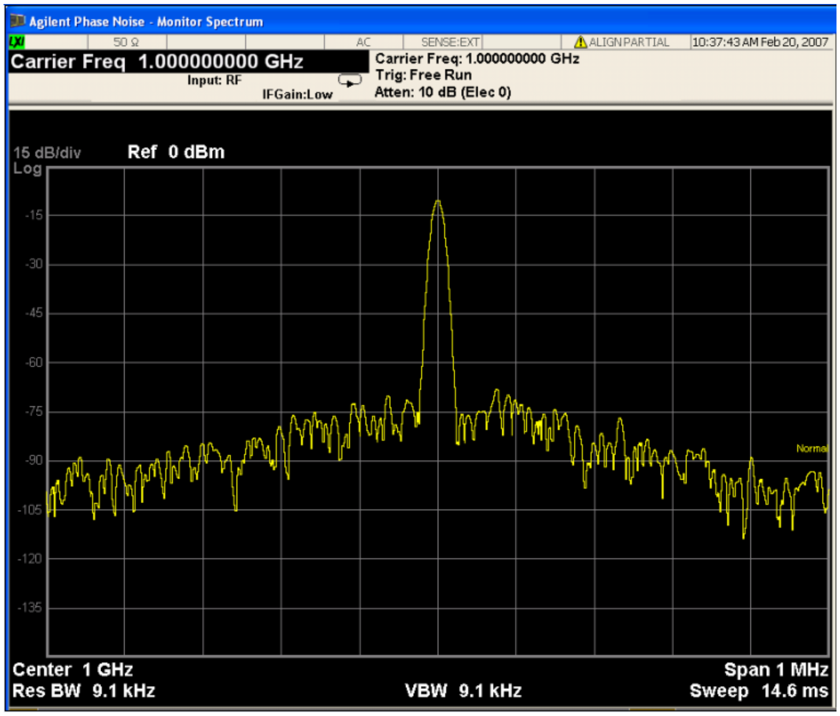
NOTE A display with a Spectrum window appears when you activate a Spectrum measurement. Changes to the **FREQ**, **Span**, or **AMPTD** settings will affect only the active window.

The default display shows the **Current** (yellow trace) data. To make viewing the display easier, you can view either the **Current** trace or

Average separately.

- Press **Trace/Detector, Select Trace** and select the trace(s) desired for display, then toggle **Display** to **Show**.

Figure 12-3 Monitor Spectrum Measurement - Spectrum and I/Q Waveform (Default View)



*Meas Setup: Span = 5.000 MHz,
Others = Factory default settings

*Input signals: –10.00 dBm, Test Model 1

Step 8. To make a measurement repeatedly, press **Cont**.

13 IQ Waveform (Time Domain) Measurements

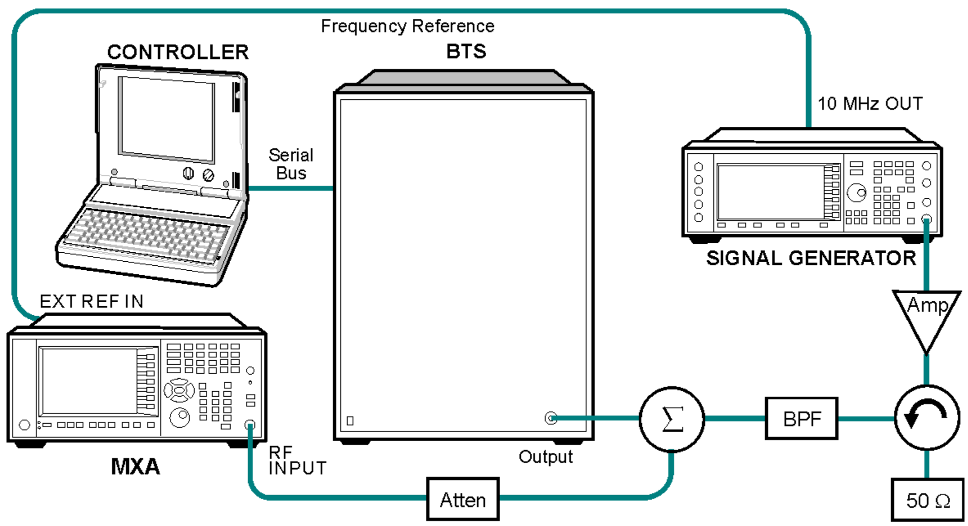
This chapter explains how to make a waveform (time domain) measurement on a W-CDMA base transceiver station (BTS). The measurement of I and Q modulated waveforms in the time domain disclose the voltages which comprise the complex modulated waveform of a digital signal.

Setting Up and Making Measurements

Configuring the Measurement System

The BTS under test must be set to transmit the RF power remotely through the system controller. This transmitting signal is connected to the RF input port of the instrument. Connect the equipment as shown. An interfering or adjacent signal may supplied as shown.

Figure 13-1 Waveform Measurement System



1. Using the appropriate cables, adapters, and circulator, connect the output signal of the BTS to the RF input of the analyzer.
2. Connect the base transmission station simulator or signal generator to the BTS through a circulator to initiate a link constructed with sync and pilot channels, if required.
3. Connect a BNC cable between the 10 MHz OUT port of the signal generator and the EXT REF IN port of the analyzer.
4. Connect the system controller to the BTS through the serial bus cable to control the BTS operation.
5. Connect an external trigger, if needed. Press **Mode Setup**, **Trigger** to access a menu to set up inputs and levels for all triggers. You must then select the appropriate trigger under the **Meas Setup**, **Trigger** menu to direct the measurement to use your trigger settings.

Setting the BTS

From the base transceiver station simulator and the system controller, set up a call using loopback mode for the BTS to transmit the RF signal.

Measurement Procedure

Step 1. Set the analyzer to the appropriate mode and enable the W-CDMA Mode measurements:

Press **Mode**, W-CDMA with HSDPA/HSUPA.

Step 2. Preset the analyzer mode:

Press **Mode Preset**.

Step 3. Set the measurement center frequency:

Press **FREQ Channel**, enter a numerical frequency using the front-panel keypad, and select a units key, for example **MHz**.

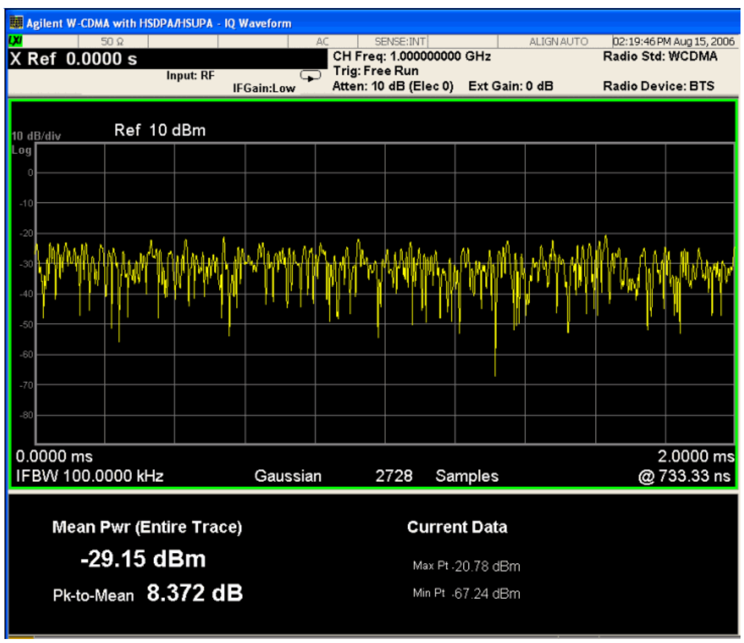
Step 4. Set the measurement span frequency:

Press **SPAN X Scale** key, enter a numerical span using the front-panel keypad, and select a units key, for example **MHz**.

Step 5. Initiate the IQ Waveform measurement:

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

Figure 13-2 Waveform Measurement - RF Envelope (Default View)



*Meas Setup: View/Display = RF Envelope View,
Others = Factory default settings

*Input signal: W-CDMA (3GPP 3.4 12-00), Test Model 1,

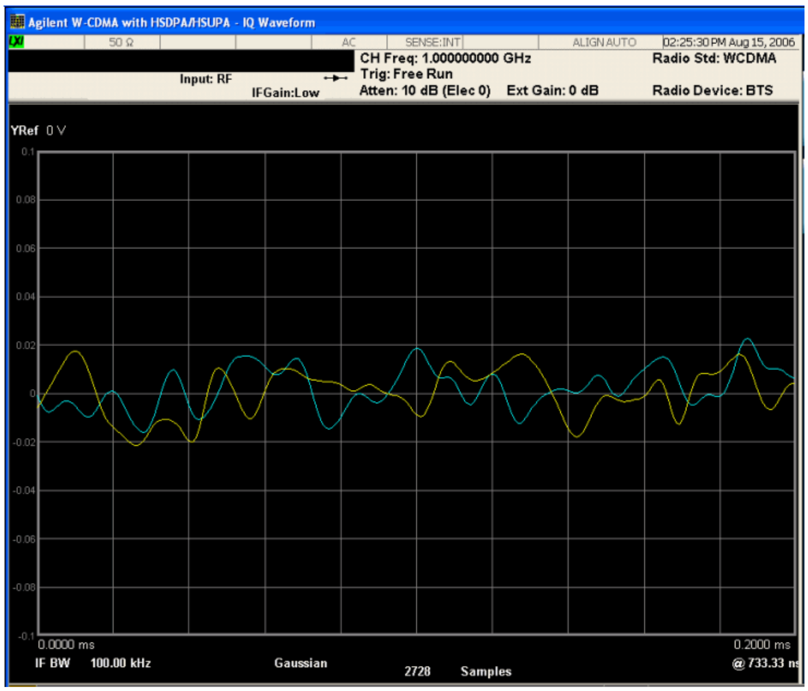
The default display shows the RF Envelope with the current data. The measured values for the mean power and peak-to-mean power are shown in the text window.

Step 6. Select the IQ Waveform view.

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

- The IQ Waveform window provides a view of the I and Q waveforms together on the same graph in terms of voltage versus time in linear scale.

Figure 13-3 Waveform Measurement- IQ Waveform View



*Meas Setup: View/Display = IQ Waveform View,
Others = Factory defaults, except X and Y scales

*Input signal: W-CDMA (3GPP 3.4 12-00), Test Model 1,

Step 7. Press the **Marker**, **Select Marker**, keys to activate a marker. Rotate the RPG knob until the marker is shown at a desired time in the waveform for viewing the trace values at the time position of the marker.

Step 8. To make a measurement repeatedly, press **Cont**.

Step 9. Press the **Meas Setup** key to see the keys available to change the measurement parameters from the default condition.

Using the Waveform Measurement to Set Up Triggering (for burst signals)

You can use the waveform measurement to view your signal in the time domain and to help select the appropriate trigger to acquire your signal.

Step 1. Activate the waveform measurement view:

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

Step 2. Adjust the scale of the x-axis to view the complete signal waveform:

Press **SPAN X Scale**, **Scale/Div**, then use the front-panel keypad to input the scale/div, then press a units key, for example μ**s**, to complete the entry.

Step 3. Select a trigger source (free run is the default setting):

Press **Trigger**, then select one of the available trigger sources. You can also setup trigger holdoff and auto trigger timing.

14 Programming Examples

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

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

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

Visual Basic is a registered trademark of Microsoft Corporation.

Available Programing Examples

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

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

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

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

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

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

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

  File name: _screen.bas

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

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

  File name: bintrace.bas

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

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

  This example demonstrates how to:

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

File name: _Sweep.c

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

  This example demonstrates how:

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

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

  File name: _SRQ.C

- *Relative Band Power Markers* using C++

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

  File name: _BPM.c

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

  This example demonstrates how to:

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

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

  File name: _tracecouple.c

- *Phase Noise* using C++

  This example demonstrates how to:

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

  File name: _phasenoise.c

Programming using Agilent VEE Pro:

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

  This example program stores the current screen image on the instrument flash memory as "`D:\scr.png`". It then transfers the image over GPIB and stores the image on your PC in the desired directory as "`capture.gif`". The file "`D:\scr.png`" is then deleted on the instrument flash memory.

  File name: _ScreenCapture.vee

- *Transfer Trace Data* data transfer using Agilent VEE Pro

  This example program transfers the trace data from your Spectrum Analyzer. The program queries the IDN string from the instrument and supports Integer 32, real 32, real 64 and ASCII data. The program returns 1001 trace points for the signal analyzer.

  File name: transfertrace.vee

Programming Fundamentals

SCPI Language Basics

This section is not intended to teach you everything about the SCPI (Standard Commands for Programmable Instruments) programming language. The SCPI Consortium or IEEE can provide that level of detailed information. For more information refer to the websites for the IEEE Standard 488.1 (IEEE Standard Digital Interface for Programmable Instrumentation).

Topics covered in this chapter include:

- "Command Keywords and Syntax" on page 78
- "Creating Valid Commands" on page 78
- "Special Characters in Commands" on page 79
- "Parameters in Commands" on page 80
- "Putting Multiple Commands on the Same Line" on page 83

Command Keywords and Syntax

A typical command is made up of keywords set off by colons. The keywords are followed by parameters that can be followed by optional units.

Example: `SENSe:FREQuency:STARt 1.5 MHZ`

The instrument does not distinguish between upper and lower case letters. In the documentation, upper case letters indicate the short form of the keyword. The lower case letters, indicate the long form of the keyword. Either form may be used in the command.

Example: `Sens:Freq:Star 1.5 mhz`

is the same as `SENSE:FREQ:start 1.5 MHz`

NOTE The command `SENS:FREQU:STAR` would not be valid because `FREQU` is neither the short, nor the long form of the command. Only the short and long forms of the keywords are allowed in valid commands.

Creating Valid Commands

Commands are not case sensitive and there are often many different ways of writing a particular command. These are examples of valid commands for a given command syntax:

Command Syntax	Sample Valid Commands
`[SENSe:]BANDwidth[:RESolution] <freq>`	The following sample commands are all identical. They will all cause the same result. • `Sense:Band:Res 1700` • `BANDWIDTH:RESOLUTION 1.7e3` • `sens:band 1.7KHZ` • `SENS:band 1.7E3Hz` • `band 1.7kHz` • `bandwidth:RES 1.7e3Hz`
`MEASure:SPECtrum[n]?`	• `MEAS:SPEC?` • `Meas:spec?` • `meas:spec3?` The number 3 in the last meas example causes it to return different results then the commands above it. See the command description for more information.
`[:SENSe]:DETector[:FUNCtion] NEGative|POSitive|SAMPle`	• `DET:FUNC neg` • `Detector:Func Pos`
`INITiate:CONTinuous ON|OFF|1|0`	The sample commands below are identical. • `INIT:CONT ON` • `init:continuous 1`

Special Characters in Commands

Special Character	Meaning	Example
|	A vertical stroke between **parameters** indicates alternative choices. The effect of the command is different depending on which parameter is selected.	Command: `TRIGger:SOURce EXTernal|INTernal|LINE` The choices are external, internal, and line. Ex: `TRIG:SOURCE INT` is one possible command choice.
	A vertical stroke between **keywords** indicates identical effects exist for both keywords. The command functions the same for either keyword. Only one of these keywords is used at a time.	Command: `SENSe:BANDwidth|BWIDth:OFFSet` Two identical commands are: Ex1: `SENSE:BWIDTH:OFFSET` Ex2: `SENSE:BAND:OFFSET`

Special Character	Meaning	Example
[ ]	Keywords in square brackets are optional when composing the command. These implied keywords will be executed even if they are omitted.	Command: `[SENSe:]BANDwidth[:RESolution]:AUTO` The following commands are all valid and have identical effects: Ex1: `bandwidth:auto` Ex2: `band:resolution:auto` Ex3: `sense:bandwidth:auto`
< >	Angle brackets around a word, or words, indicates they are not to be used literally in the command. They represent the needed item.	Command: `SENS:FREQ <freq>` In this command example the word <freq> should be replaced by an actual frequency. Ex: `SENS:FREQ 9.7MHz`.
{ }	Parameters in braces can optionally be used in the command either not at all, once, or several times.	Command: `MEASure:BW <freq>{,level}` A valid command is: `meas:BW 6MHz, 3dB, 60dB`

Parameters in Commands

There are four basic types of parameters: booleans, keywords, variables and arbitrary block program data.

OFF|ON|0|1

(Boolean) This is a two state boolean-type parameter. The numeric value 0 is equivalent to OFF. Any numeric value other than 0 is equivalent to ON. The numeric values of 0 or 1 are commonly used in the command instead of OFF or ON. Queries of the parameter always return a numeric value of 0 or 1.

keyword The keywords that are allowed for a particular command are defined in the command syntax description.

Units Numeric variables may include units. The valid units for a command depend on the variable type being used. See the following variable descriptions. The indicated default units will be used if no units are sent. Units can follow the numerical value with, or without, a space.

Variable A variable can be entered in exponential format as well as standard numeric format. The appropriate range of the variable and its optional units are defined in the command description.

The following keywords may also be used in commands, but not all commands allow keyword variables.

- DEFault - resets the parameter to its default value.
- UP - increments the parameter.
- DOWN - decrements the parameter.
- MINimum - sets the parameter to the smallest possible value.
- MAXimum - sets the parameter to the largest possible value.

The numeric value for the function’s MINimum, MAXimum, or DEFault can be queried by adding the keyword to the command in its query form. The keyword must be entered following the question mark.

Example query: `SENSE:FREQ:CENTER? MAX`

Variable Parameters

<integer> is an integer value with no units.

<real> Is a floating point number with no units.

<freq>
<bandwidth> Is a positive rational number followed by optional units. The default unit is Hertz. Acceptable units include: Hz, kHz, MHz, GHz.

<time>
<seconds> Is a rational number followed by optional units. The default units are seconds. Acceptable units include: ks, s, ms, µs, ns.

<voltage> Is a rational number followed by optional units. The default units are Volts. Acceptable units include: V, mV, µV, nV

<current> Is a rational number followed by optional units. The default units are Amperes. Acceptable units include: A, mA, µA, nA.

<power> Is a rational number followed by optional units. The default units are W. Acceptable units include: mAW, kW, W, mW, µW, nW, pW.

<ampl> Is a rational number followed by optional units. The default units are dBm. Acceptable units include: dBm, dBmV, dBµV.

<rel_power>
<rel_ampl> Is a positive rational number followed by optional units. The default units are dB. Acceptable units include: dB.

<percent> Is a rational number between 0 and 100. You can either use no units or use PCT.

<angle>
<degrees> Is a rational number followed by optional units. The default units are degrees. Acceptable units include: DEG, RAD.

<string> Is a series of alpha numeric characters.

<bit_pattern> Specifies a series of bits rather than a numeric value. The bit series is the binary representation of a numeric value. There are no units.

Bit patterns are most often specified as hexadecimal numbers, though octal, binary or decimal numbers may also be used. In the SCPI language these numbers are specified as:

- Hexadecimal, #Hdddd or #hdddd where 'd' represents a hexadecimal digit 0 to 9 and 'a' to 'f'. So #h14 can be used instead of the decimal number 20.
- Octal, #Odddddd or #odddddd where 'd' represents an octal digit 0 to 7. So #o24 can be used instead of the decimal number 20.
- Binary, #Bdddddddddddddddd or #bdddddddddddddddd where 'd' represents a 1 or 0. So #b10100 can be used instead of the decimal number 20.

Block Program Data

Some parameters consist of a block of data. There are a few standard types of block data. Arbitrary blocks of program data can also be used.

<trace> Is an array of rational numbers corresponding to displayed trace data. See FORMat:DATA for information about available data formats.

A SCPI command often refers to a block of current trace data with a variable name such as: Trace1, Trace2, or Trace3, depending on which trace is being accessed.

<arbitrary block data> Consists of a block of data bytes. The first information sent in the block is an ASCII header beginning with #. The block is terminated with a semi-colon. The header can be used to determine how many bytes are in the data block. There are no units. You will not get block data if your data type is ASCII, using `FORMat:DATA ASCII` command. Your data will be comma separated ASCII values.

Block data example: suppose the header is #512320.

- The first digit in the header (5) tells you how many additional digits/bytes there are in the header.

- The 12320 means 12 thousand, 3 hundred, 20 data bytes follow the header.
- Divide this number of bytes by your current data format (bytes/data point), either 8 (for real,64), or 4 (for real,32). For this example, if you are using real64 then there are 1540 points in the block.

Putting Multiple Commands on the Same Line

Multiple commands can be written on the same line, reducing your code space requirement. To do this:

- Commands must be separated with a semicolon (;).
- If the commands are in different subsystems, the key word for the new subsystem must be preceded by a colon (:).
- If the commands are in the same subsystem, the full hierarchy of the command key words need not be included. The second command can start at the same key word level as the command that was just executed.

SCPI Termination and Separator Syntax

All binary trace and response data is terminated with <NL><END>, as defined in Section 8.5 of IEEE Standard 488.2-1992, *IEEE Standard Codes, Formats, Protocols and Common Commands for Use with ANSI/IEEE Std 488.1-1987*. New York, NY, 1992. (Although one intent of SCPI is to be interface independent, <END> is only defined for IEEE 488 operation.)

The following are some examples of good and bad commands. The examples are created from a theoretical instrument with the simple set of commands indicated below:

```
[:SENSe]
     :POWer
          [:RF]
          :ATTenuation 40dB

:TRIGger
     [:SEQuence]
     :EXTernal [1]
          :SLOPe
               POSitive

[:SENSe]
     :FREQuency
          :STARt
     :POWer
     [:RF]
          :MIXer
               :RANGe
              [:UPPer]
```


Programming Examples

Bad Command	Good Command
PWR:ATT 40dB	POW:ATT 40dB
The short form of POWER is POW, not PWR.	
FREQ:STAR 30MHz;MIX:RANG -20dBm	FREQ:STAR 30MHz;POW:MIX:RANG -20dBm
The MIX:RANG command is in the same :SENSE subsystem as FREQ, but executing the FREQ command puts you back at the SENSE level. You must specify POW to get to the MIX:RANG command.	
FREQ:STAR 30MHz;POW:MIX RANG -20dBm	FREQ:STAR 30MHz;POW:MIX:RANG -20dBm
MIX and RANG require a colon to separate them.	
:POW:ATT 40dB;TRIG:FREQ:STAR 2.3GHz	:POW:ATT 40dB;:FREQ:STAR 2.3GHz
:FREQ:STAR is in the :SENSE subsystem, not the :TRIGGER subsystem.	
:POW:ATT?:FREQ:STAR?	:POW:ATT?;:FREQ:STAR?
:POW and FREQ are within the same :SENSE subsystem, but they are two separate commands, so they should be separated with a semicolon, not a colon.	
:POW:ATT -5dB;:FREQ:STAR 10MHz	:POW:ATT 5dB;:FREQ:STAR 10MHz
Attenuation cannot be a negative value.	

Improving Measurement Speed

There are a number of things you can do in your programs to make them run faster:

Turn off the display updates

`:DISPlay:ENABle OFF` turns off the display. That is, the data may still be visible, but it will no longer be updated. Updating the display slows down the measurement. For remote testing, since the computer is processing the data rather than a person, there is no need to display the data on the analyzer screen.

Use binary data format instead of ASCII

The ASCII data format is the instrument default since it is easier for people to understand and is required by SCPI for *RST. However, data input/output is faster using the binary formats.

`:FORMat:DATA REAL,64` selects the 64-bit binary data format for all your numerical data queries. You may need to swap the byte order if you are using a PC rather than UNIX. **`NORMal`** is the default byte order. Use **`:FORMat:BORDer SWAP`** to change the byte order so that the least significant byte is sent first. (Real,32 which is smaller and somewhat faster, should only be used if you do not need full resolution for your data. Some frequency data may require full 64-bit resolution.)

When using the binary format, data is sent in a block of bytes with an ASCII header. A data query would return the block of data in the following format: `#DNNN<nnn binary data bytes>`

To parse the data:

- Read two characters (#D), where D tells you how many N characters follow the D character.
- Read D characters, the resulting integer specifies the number of data bytes sent.
- Read the bytes into a real array.

For example, suppose the header is #512320.

- The first character/digit in the header (5) tells you how many

Programming Examples

additional digits there are in the header.

- The 12320 means 12 thousand, 3 hundred, 20 data bytes follow the header.
- Divide this number of bytes by your current data format (bytes/data point), 8 for real,64. For this example, there are 1540 data points in the block of data.

Minimize the number of GPIB transactions

When you are using the GPIB for control of your instrument, each transaction requires driver overhead and bus handshaking, so minimizing these transactions reduces the time used.

- You can reduce bus transactions by sending multiple commands per transaction. See the information on "Putting Multiple Commands on the Same Line" in the SCPI Language Basics section.
- If you are making the same measurement multiple times with small changes in the measurement setup, use the READ command. It is faster then using INITiate and FETCh.

Consider using USB or LAN instead of GPIB

USB and LAN allow faster data input and output. This is especially important if you are moving large blocks of data. You will not get this improved throughput using LAN if there is excessive LAN traffic (that is, your test instrument is connected to a very busy enterprise LAN). You may want to use a private LAN that is only for your test system.

Minimize DUT/instrument setup changes

- Some instrument setup parameters are common to multiple measurements. You should look at your measurement process with an eye toward minimizing setup changes. If your test process involves nested loops, make sure that the inner-most loop is the fastest. Also, check if the loops could be nested in a different order to reduce the number of parameter changes as you step through the test.
- Are you are using the measurements under the **Meas** key? Remember that if you have already set your Meas Setup parameters for a measurement, and you want to make another one of these measurements later, use READ:<meas>?. The MEASure:<meas>?. command resets all the settings to the defaults, while READ changes back to that measurement without changing the setup parameters from the previous use.
- Are you are using the Measurements under the **Meas** key? Remember that *Mode Setup* parameters remain constant across all the measurements in that mode (for example, center/channel frequency, amplitude, radio standard, input selection, trigger setup). You do not have to re-initialize them each time you change to a

different measurement.

Avoid unnecessary use of *RST

Remember that while *RST does not change the current Mode, it presets all the measurements and settings to their factory defaults. This forces you to reset your analyzer's measurement settings even if they use similar mode setup or measurement settings. See Minimize DUT/instrument setup changes. (Also note that *RST may put the instrument in single measurement/sweep for some modes.)

Avoid automatic attenuator setting

Many of the one-button measurements use an internal process for automatically setting the value of the attenuator. It requires measuring an initial burst to identify the proper attenuator setting before the next burst can be measured properly. If you know the amount of attenuation or the signal level needed for your measurement, just set it.

Note that spurious types of measurements must be done with the attenuator set to automatic (for measurements like: output RF spectrum, transmit spurs, adjacent channel power, spectrum emission mask). These types of measurements start by tuning to the signal, then they tune away from it and must be able to reset the attenuation value as needed.

Avoid using RFBurst trigger for single burst signals

RFBurst triggering works best when measuring signals with repetitive bursts. For a non-repetitive or single burst signals, use the IF (video) trigger or external trigger, depending on what you have available.

RFBurst triggering depends on its establishment of a valid triggering reference level, based on previous bursts. If you only have a single burst, the peak detection nature of this triggering function, may result in the trigger being done at the wrong level/point generating incorrect data, or it may not trigger at all.

Are you making a single burst measurement? To get consistent triggering and good data for this type of measurement application, you need to synchronize the triggering of the DUT with the analyzer. You should use the analyzer's internal status system for this.

The first step in this process is to initialize the status register mask to look for the "waiting for trigger" condition (bit 5). Use `:STATus:OPERation:ENABle 32`

Then, in the measurement loop:

1. `:STATus:OPERation:EVENt?` This query of the operation event register is to clear the current register contents.
2. `:READ:PVT?` initiates a measurement (in this example, for GSM

power versus time) using the previous setup. The measurement will then be waiting for the trigger.

Make sure the attenuation is set manually. Do NOT use automatic attenuation as this requires an additional burst to determine the proper attenuation level before the measurement can be made.

3. Create a small loop that will serial poll the instrument for a status byte value of binary 128. Then wait 1 msec (100 ms if the display is left on/enabled) before checking again, to keep the bus traffic down. These two commands are repeated until the condition is set, so we know that the trigger is armed and ready.
4. Trigger your DUT to send the burst.
5. Return the measurement data to your computer.

NOTE This process cannot be done with the current VXI plug-n-play driver implementation. You will need to use the previous SCPI commands.

Programming in C Using the VTL

The programming examples that are provided are written using the C programming language and the Agilent VTL (VISA transition library). This section includes some basic information about programming in the C language. Note that some of this information may not be relevant to your particular application. (For example, if you are not using VXI instruments, the VXI references will not be relevant).

Refer to your C programming language documentation for more details. (This information is taken from the manual "VISA Transition Library", part number E2090-90026.) The following topics are included:

Typical Example Program Contents

The following is a summary of the VTL function calls used in the example programs.

`visa.h` This file is included at the beginning of the file to provide the function prototypes and constants defined by VTL.

`ViSession` The `ViSession` is a VTL data type. Each object that will establish a communication channel must be defined as `ViSession`.

`viOpenDefaultRM` You must first open a session with the default resource manager with the `viOpenDefaultRM` function. This function will initialize the default resource manager and return a pointer to that resource manager session.

`viOpen` This function establishes a communication channel with the device specified. A session identifier that can be used with other VTL functions is returned. This call must be made for each device you will be using.

`viPrintf`
`viScanf` These are the VTL formatted I/O functions that are patterned after those used in the C programming language. The `viPrintf` call sends the IEEE 488.2 `*RST` command to the instrument and puts it in a known state. The `viPrintf` call is used again to query

for the device identification (*IDN?). The viScanf call is then used to read the results.

viClose — This function must be used to close each session. When you close a device session, all data structures that had been allocated for the session will be de-allocated. When you close the default manager session, all sessions opened using the default manager session will be closed.

Linking to VTL Libraries

Your application must link to one of the VTL import libraries:

32-bit Version:

C:\VXIPNP\WIN95\LIB\MSC\VISA32.LIB for Microsoft compilers

C:\VXIPNP\WIN95\LIB\BC\VISA32.LIB for Borland compilers

16-bit Version:

C:\VXIPNP\WIN\LIB\MSC\VISA.LIB for Microsoft compilers

C:\VXIPNP\WIN\LIB\BC\VISA.LIB for Borland compilers

See the following section, “Compiling and Linking a VTL Program” for information on how to use the VTL run-time libraries.

Compiling and Linking a VTL Program

32-bit Applications

The following is a summary of important compiler-specific considerations for several C/C++ compiler products when developing WIN32 applications.

For Microsoft Visual C++ version 2.0 compilers:

- Select Project | Update All Dependencies from the menu.
- Select Project | Settings from the menu. Click on the C/C++ button. Select Code Generation from the Use Run-Time Libraries list box. VTL requires these definitions for WIN32. Click OK to close the dialog boxes.
- Select Project | Settings from the menu. Click on the Link button and add visa32.lib to the Object / Library Modules list box. Optionally, you may add the library directly to your project file. Click OK to close the dialog boxes.
- You may wish to add the include file and library file search paths. They are set by doing the following:
  1. Select Tools | Options from the menu.
  2. Click Directories to set the include file path.

3. Select Include Files from the Show Directories For list box.
4. Click Add and type in the following:
   C:\VXIPNP\WIN95\INCLUDE
5. Select Library Files from the Show Directories For list box.
6. Click Add and type in the following:
   C:\VXIPNP\WIN95\LIB\MSC

For Borland C++ version 4.0 compilers:

- You may wish to add the include file and library file search paths. They are set under the Options | Project menu selection. Double-click on Directories from the Topics list box and add the following:

  C:\VXIPNP\WIN95\INCLUDE
  C:\VXIPNP\WIN95\LIB\BC

16-bit Applications

The following is a summary of important compiler-specific considerations for the Windows compiler.

For Microsoft Visual C++ version 1.5:

- To set the memory model, do the following:
  1. Select Options | Project.
  2. Click Compiler, then select Memory Model from the Category list.
  3. Click the Model list arrow to display the model options, and select Large.
  4. Click OK to close the Compiler dialog box.
- You may wish to add the include file and library file search paths. They are set under the Options | Directories menu selection:

  C:\VXIPNP\WIN\INCLUDE
  C:\VXIPNP\WIN\LIB\MSC

  Otherwise, the library and include files should be explicitly specified in the project file.

Example Program

This example program queries a GPIB device for an identification string and prints the results. Note that you must change the address.

```
/*idn.c - program filename */

#include "visa.h"
```

```
#include <stdio.h>

void main ()
{
     /*Open session to GPIB device at address 18 */
     ViOpenDefaultRM (&defaultRM);
     ViOpen (defaultRM, GPIB0::18::INSTR", VI_NULL,
       VI_NULL, &vi);

     /*Initialize device */
     viPrintf (vi, "*RST\n");

     /*Send an *IDN? string to the device */
     printf (vi, "*IDN?\n");

     /*Read results */
     viScanf (vi, "%t", &buf);

     /*Print results */
     printf ("Instrument identification string: %s\n", buf);

     /* Close sessions */
     viClose (vi);
     viClose (defaultRM);
}
```

Including the VISA Declarations File

For C and C++ programs, you must include the `visa.h` header file at the beginning of every file that contains VTL function calls:

```
#include "visa.h"
```

This header file contains the VISA function prototypes and the definitions for all VISA constants and error codes. The `visa.h` header file includes the `visatype.h` header file.

The `visatype.h` header file defines most of the VISA types. The VISA types are used throughout VTL to specify data types used in the functions. For example, the `viOpenDefaultRM` function requires a pointer to a parameter of type `ViSession`. If you find `ViSession` in the `visatype.h` header file, you will find that `ViSession` is eventually typed as an unsigned long.

Opening a Session

A session is a channel of communication. Sessions must first be opened on the default resource manager, and then for each device you will be using. The following is a summary of sessions that can be opened:

- A **resource manager session** is used to initialize the VISA system. It is a parent session that knows about all the opened sessions. A resource manager session must be opened before any other session can be opened.

Programming Examples

- A **device session** is used to communicate with a device on an interface. A device session must be opened for each device you will be using. When you use a device session you can communicate without worrying about the type of interface to which it is connected. This insulation makes applications more robust and portable across interfaces. Typically a device is an instrument, but could be a computer, a plotter, or a printer.

NOTE All devices that you will be using need to be connected and in working condition prior to the first VTL function call (`viOpenDefaultRM`). The system is configured only on the *first* `viOpenDefaultRM` per process. Therefore, if `viOpenDefaultRM` is called without devices connected and then called again when devices are connected, the devices will not be recognized. You must close **ALL** resource manager sessions and re-open with all devices connected and in working condition.

Device Sessions

There are two parts to opening a communications session with a specific device. First you must open a session to the default resource manager with the `viOpenDefaultRM` function. The first call to this function initializes the default resource manager and returns a session to that resource manager session. You only need to open the default manager session once. However, subsequent calls to `viOpenDefaultRM` returns a session to a unique session to the same default resource manager resource.

Next, you open a session with a specific device with the `viOpen` function. This function uses the session returned from `viOpenDefaultRM` and returns its own session to identify the device session. The following shows the function syntax:

viOpenDefaultRM (*sesn*);

viOpen (*sesn*, *rsrcName*, *accessMode*, *timeout*, *vi*);

The session returned from `viOpenDefaultRM` must be used in the *sesn* parameter of the `viOpen` function. The `viOpen` function then uses that session and the device address specified in the *rsrcName* parameter to open a device session. The *vi* parameter in `viOpen` returns a session identifier that can be used with other VTL functions.

Your program may have several sessions open at the same time by creating multiple session identifiers by calling the `viOpen` function multiple times.

The following summarizes the parameters in the previous function calls:

sesn This is a session returned from the `viOpenDefaultRM` function that identifies the resource manager session.

rsrcName	This is a unique symbolic name of the device (device address).
accessMode	This parameter is not used for VTL. Use VI_NULL.
timeout	This parameter is not used for VTL. Use VI_NULL.
vi	This is a pointer to the session identifier for this particular device session. This pointer will be used to identify this device session when using other VTL functions.

The following is an example of opening sessions with a GPIB multimeter and a GPIB-VXI scanner:

```
ViSession defaultRM, dmm, scanner;
.
.
viOpenDefaultRM(&defaultRM);
viOpen (defaultRM, "GPIB0::22::INSTR", VI_NULL,
    VI_NULL, &dmm);
viOpen (defaultRM, "GPIB-VXI0::24::INSTR", VI_NULL,
    VI_NULL, &scanner);
.
.
viClose (scanner);
viClose (dmm);
viClose(defaultRM);
```

The above function first opens a session with the default resource manager. The session returned from the resource manager and a device address is then used to open a session with the GPIB device at address 22. That session will now be identified as **dmm** when using other VTL functions. The session returned from the resource manager is then used again with another device address to open a session with the GPIB-VXI device at primary address 9 and VXI logical address 24. That session will now be identified as **scanner** when using other VTL functions. See the following section for information on addressing particular devices.

Addressing a Session

As seen in the previous section, the *rsrcName* parameter in the `viOpen` function is used to identify a specific device. This parameter is made up of the VTL interface name and the device address. The interface name is determined when you run the VTL Configuration Utility. This name is usually the interface type followed by a number. The following table illustrates the format of the *rsrcName* for the different interface types

Interface	**Syntax**
VXI	VXI [*board*]::*VXI logical address*[::INSTR]
GPIB-VXI	GPIB-VXI [*board*]::*VXI logical address*[::INSTR]

Interface	Syntax
GPIB	GPIB [*board*]::*primary address*[::*secondary address*][::INSTR]

The following describes the parameters used above:

board — This optional parameter is used if you have more than one interface of the same type. The default value for *board* is 0.

VSI logical address — This is the logical address of the VXI instrument.

primary address — This is the primary address of the GPIB device.

secondary address — This optional parameter is the secondary address of the GPIB device. If no secondary address is specified, none is assumed.

INSTR — This is an optional parameter that indicates that you are communicating with a resource that is of type **INSTR**, meaning instrument.

NOTE If you want to be compatible with future releases of VTL and VISA, you must include the INSTR parameter in the syntax.

The following are examples of valid symbolic names:

XI0::24::INSTR — Device at VXI logical address 24 that is of VISA type INSTR.

VXI2::128 — Device at VXI logical address 128, in the third VXI system (VXI2).

GPIB-VXI0::24 — A VXI device at logical address 24. This VXI device is connected via a GPIB-VXI command module.

GPIB0::7::0 — A GPIB device at primary address 7 and secondary address 0 on the GPIB interface.

The following is an example of opening a device session with the GPIB device at primary address23.

```
ViSession defaultRM, vi;

.
.

viOpenDefaultRM (&defaultRM);

viOpen (defaultRM, "GPIB0::23::INSTR", VI_NULL,VI_NULL,&vi);

.
.
```

Programming Examples

```
viClose(vi);

viClose (defaultRM);
```

Closing a Session

The `viClose` function must be used to close each session. You can close the specific device session, which will free all data structures that had been allocated for the session. If you close the default resource manager session, all sessions opened using that resource manager will be closed.

Since system resources are also used when searching for resources (`viFindRsrc`) or waiting for events (`viWaitOnEvent`), the `viClose` function needs to be called to free up find lists and event contexts.

Index