Visual System Simulator™ 2007

Measurement Reference

Visual System Simulator Measurement Reference

Version 7.5, October 2007

Applied Wave Research, Inc.
1960 E. Grand Avenue, Suite 430
El Segundo, CA 90245

Phone 310.726.3000
Fax 310.726.3005
Website *www.appwave.com*
Technical Support Email *support@appwave.com*
Technical Support Phone 888.349.7610

CONTENTS

PREFACE

Visual System Simulator™ (VSS) is an electronic design automation tool that provides an interactive solution for modeling, analyzing, and optimizing analog and digital communication systems. VSS integrates radio frequency (RF) and digital signal processing (DSP) analysis to provide an end-to-end communications systems design solution.

ABOUT THIS BOOK

This book provides complete reference information on all of the measurements that you can choose as output for your system simulations.

The measurements in this guide are organized alphabetically in categories such as "System" and "System BER". This organization reflects how they are displayed in VSS. (They are displayed under "System" in **Meas. Type** and under sub-classes such as BER and NW Analyzer in **Measurement** in the Add Measurement dialog box.)

For each measurement, the following attributes are described:

Attribute	Description
Measurement Type	The general category of measurement, such as "System" or "System Spectrum" is indicated in the header of each measurement document.
Summary	Describes the measurement and provides information on what this measurement is typically used for.
Parameters	The user-modifiable input parameters for this measurement.
Result	Specifies the format of value returned by the simulator, such as a real value or a complex value, and the relevant axis units.
Graph Types	Type of graph, such as constellation, on which this measurement can be charted.

Attribute	Description
Computational Details	Additional details about how the measurement is computed
Options	If applicable, indicates the post-processing (such as smoothing) available for this measurement.

This guide assumes that you have a working knowledge of high-frequency electronic design, layout, and analysis.

Additional Documentation

The Visual System Simulator product includes the following additional documentation:

- *What's New in VSS 2007?* presents the new features, system blocks, and measurements for this release.
- *MWO/VSS/AO Installation Guide* (available on your Program Disk (as *install.pdf*) or downloadable from the Applied Wave Research website at *www.appwave.com* under Support) describes how to install the AWR® Design Environment™ suite and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips.
- *MWO/VSS/AO Getting Started Guide* includes a quick installation procedure and familiarizes you with the AWR Design Environment through MWO, VSS, and AO example sections. Microwave Office example projects show how to design and analyze simple linear, nonlinear, and EM circuits, and how to create layouts. Visual System Simulator examples show how to design systems and perform simulations using predefined or customized transmitters and receivers. Analog Office examples show how to design circuits composed of schematics and electromagnetic (EM) structures from an extensive electrical model set, and then generate physical layouts of the designs. You can perform simulations using a number of simulators, and then display the output in a wide variety of graphical forms based on your analysis needs. You can also tune or optimize the designs, and your changes are automatically and immediately reflected in the layout.
- *Visual System Simulator System Block Catalog* provides complete reference information on all of the system blocks that you use to build systems.
- *VSS Modeling Guide* contains information on simulation basics, RF modeling capabilities, and noise modeling.

This guide uses the following typographical conventions.

Item	Convention
Anything that you select (or click on) in the VSS design environment, like menus, submenus, menu items, dialog box options, button names, and icon names	Shown in a bold type. Nested menu selections are shown with a "**>**" to indicate that you select the first menu item and then select the second menu item from the menu: Select **File > New Project**.
Text that you enter using the keyboard	Shown in a bold type between quotes: Enter "**my_project**" in **Project Name**.
Keys or key combinations that you press	Shown in a bold type with initial capitals. Key combinations using a "+" indicate that you press and hold the first key while pressing the second key: Press **Alt+F1**.
Filenames, directory names, websites, and e-mail addresses	Shown in italics: Contact *support@appwave.com* for more information.
Any field within a file	Shown in an alternate bold type: Define this parameter in the **$DEFAULT_VALUES** field.
Directory in which VSS is installed	VSS program directory.

GETTING ON-LINE HELP

VSS online Help provides comprehensive information about VSS windows, menu items, dialog boxes, and design concepts.

To access context-sensitive Help for each measurement:

- After creating a graph, select **Project > Add Measurement**. Select the measurement type and measurement of interest from **Meas. Type** and **Measurement**, and then click the **Meas. Help** button.
- You can also access measurement Help using the Help table of contents by pressing the **F1** key or by choosing **Start > Programs > AWR 2007 > AWR Design Environment Help** and navigating to the VSS Measurement Reference Help book.

Annotate VSS Signal Center Frequencies: CTRFRQ

Summary

CTRFRQ displays the center frequency of complex envelope signals at output nodes of a system diagram. No output is displayed for nodes without a center frequency assigned to the signal.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in frequency units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Signal Data Rates: DRATE

Summary

DRATE displays the data rate at each output node in a system diagram. The data rate is the sampling frequency divided by the oversampling rate (samples per symbol).

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in frequency units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Static Es/N0: EsN0

Summary

EsN0 displays the static symbol energy to noise PSD property of signals at output nodes in a system diagram. No output is displayed for signals without both static signal power and static noise PSD properties.

Es/N0 is computed from the static signal power and static channel noise PSD properties of the signal. These properties can be viewed with the Annotate VSS Node Static Signal Powers annotation measurement *SIGPWR* and the Annotate VSS Node Static Noise PSDs annotation measurement *NOISEPSD*.

The ratio is computed over the signal bandwidth, which is the sampling frequency divided by the oversampling ratio.

NOTE. The static power-noise ratios are intended to be used primarily when performing BER measurements. They are used by the BER meter and measurements to determine values for the x-axis if none are explicitly specified.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in power units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Static Noise PSDs: NOISEPSD

Summary

NOISEPSD displays one of the static noise power spectral density (PSD) properties of signals at the output nodes in a system diagram. No output is displayed for signals without static noise PSD properties.

The static properties of a signal are computed prior to the generation of data samples. There are two static noise PSD properties that may be associated with a signal, channel noise PSD and thermal noise PSD.

The channel noise PSD property typically represents channel noise. Noise samples are generated and added to the signal by blocks that add channel noise, such as the Additive White Gaussian Noise block AWGN. The channel noise PSD property is used to compute signal-to-noise ratios when performing bit error rate simulations.

The thermal noise PSD property represents thermal noise. The link budget measurements use thermal noise PSD to compute cascaded noise figure.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A
Output Type	List of options	N/A

Result

This measurement returns a real value in power units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Static Phase Rotations: PHSROT

Summary

PHSROT displays the static signal phase rotation property of signals at output nodes in a system diagram. No output is displayed for signals without a static phase rotation property.

The static properties of a signal are computed prior to the generation of data samples. The static phase rotation property is an estimate of the cumulative phase rotation imparted by various blocks on the signal, as determined by the individual blocks.

In general, when estimating phase rotation a block uses the static signal power property to synthesize a sample, apply the block's transfer function to the sample, then measure the change in phase of the generated sample. The change in phase is then added to the phase rotation property of the input signal to obtain the new phase rotation property.

The static phase rotation property is normally used by the receiver blocks to derotate the signal prior to demodulation and detection. The EVM measurement also uses it to perform phase compensation.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in angle units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Static Signal Delays: SIGDLY

Summary

SIGDLY displays the static signal delay property of signals at output nodes in a system diagram. No output is displayed for signals without a static signal delay property.

The static properties of a signal are computed prior to the generation of data samples. The static signal delay property is an estimate of the signal delay imparted by various blocks on the signal, as determined by the individual blocks.

For example, FIR filter blocks introduce a signal delay equal to the position of the peak of the magnitude impulse response of the filter. Delay blocks introduce a signal delay corresponding directly to the block's delay. The static signal delay property is then the sum of all the static signal delays introduced by blocks along the signal path.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A
Output Type	List of options	N/A

Result

This measurement returns a real value. The units depend on the **Output Type** setting.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Static Signal Powers: SIGPWR

Summary

SIGPWR displays the static signal power property of signals at output nodes in a system diagram. No output displays for signals without a static signal power property.

The static properties of a signal are computed prior to the generation of data samples. The static signal power property is an estimate of the signal power generated by a power-based source block such as a transmitter or a tone source. Other blocks typically adjust the power property if they apply gain or attenuation to the signal. In general, only the RF and signal processing blocks that affect the signal power level adjust the property.

Blocks whose gain is affected by the input signal's power level, such as the nonlinear amplifiers and mixers, typically use the static power property to synthesize a single sample, apply the block's transfer function, and adjust the power property according to the power of the output sample.

The static signal power property is normally used by the receiver blocks to determine the signal gain. The EVM measurement also uses it to perform magnitude compensation. The Bit Error Rate and Symbol Error Rate blocks also use the static signal power property when automatically determining Es/N0 or Eb/N0.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in power units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Signal Sampling Frequencies: SMPFRQ

Summary

SMPFRQ displays the sampling frequency at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in frequency units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Signal Node Time Steps: TSTEP

Summary

TSTEP displays the sample time step at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Input Node Sample Backlog: SMPBKLOG

Summary

SMPBKLOG displays statistics relating to the number of unprocessed samples at each input node in a system diagram.

The statistic displayed can be the average number of samples buffered, the number of samples currently buffered, or the maximum number of samples buffered.

This measurement is useful for diagnosing simulation deadlocks and for tuning simulation speed. Simulation speed can be tuned to a certain extent by adjusting the BLKSZ parameter found in each source block or the **Data block size** default setting in the **Advanced** tab of the System Simulator Options dialog box. To tune using SMPBKLOG, adjust the block sizes to minimize the average number of samples buffered at all the nodes in the system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A
Measurement Type	List of options	N/A
Output Type	List of options	N/A

Result

This measurement returns an integer value that can be displayed as a meter, or numerically as sample counts, or as the percentage of the maximum number of samples to buffer.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Node Samples Processed Count: SMPCNT

Summary

SMPCNT displays the current number of samples processed at each node in a system diagram. For input nodes, the number of samples processed is the number of samples read from the input node by the block. For output nodes, the number of samples processed is the number of samples written to the output node by the block.

This measurement is useful when diagnosing simulation deadlocks. The point where the number of samples read by a block at its input nodes is much smaller than the number of samples written to the output nodes connected to those input nodes, is typically the location of the deadlock.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns an integer value.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Binary Format: BINFMT

Summary

BINFMT displays the binary format of the fixed-point signal at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns one of the following values:

- "2sCmp" - Two's Complement,
- "UnSgn" - Unsigned,
- "SgnMag" - Sign & Magnitude,
- "1sCmp" - One's Complement,
- "OffBin" - Binary Offset.

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Overflow Management: OVRFLOMGMT

Summary

OVRFLOMGMT displays the overflow management setting of the fixed-point signal at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns either “Truncate” or “Round-off.”

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Underflow Management: UNDRFLOMGMT

Summary

UNDRFLOMGMT displays the underflow management setting of the fixed-point signal at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement returns either “Saturate” or “Roll-over.”

Graph Type

This measurement can be displayed in a system diagram.

Annotate VSS Bit and Decimal Widths: WIDTHS

Summary

WIDTHS displays the bit width and the decimal width of the fixed-point signal at each output node in a system diagram.

Parameters

Name	Type	Range
Top Level System Diagram	Subcircuit	N/A
Results from Instance	Subcircuit	N/A

Result

This measurement displays two integer values.

Graph Type

This measurement can be displayed in a system diagram.

Instantaneous Frequency vs. Time: FREQ_INST

Summary

FREQ_INST plots instantaneous frequency versus time.

NOTE. If the measured signal is a real signal, the signal is assumed to represent phase in radians and not a voltage waveform.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Wait for Full Window	Check box	N/A
Display as Offset from Carrier	Check box	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns real values. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

If the **Wait for Full Window** setting is checked, the measurement will not generate a plot if there are fewer than the data window width of symbols available.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous frequency is computed by computing the phase difference between points and dividing by the time step:

$$f[i] = \frac{(\theta[i] - \theta[i-1] + x)}{2\pi \cdot \Delta T} = \frac{(\theta[i] - \theta[i-1] + x)}{2\pi} \cdot f_s$$

where x is used to adjust for angular wrap-around and is:

$$x = 2\pi \qquad (\theta[i] - \theta[i-1]) < \pi$$
$$x = -2\pi \qquad (\theta[i] - \theta[i-1]) > \pi$$
$$x = 0 \qquad \text{otherwise}$$

For complex signals the phase $\theta[i]$ is the phase of the complex samples. For real signals, the signal is treated as phase in radians, and $\theta[i]$ is the value of the sample.

For complex signals with non-zero center frequency, if the **Display as Offset from Carrier** settings is not checked, the center frequency is added to $f[i]$.

Inphase vs. Quadrature: IQ

Summary

IQ plots Inphase versus Quadrature of a complex waveform. The Inphase or real component is plotted along the x-axis, the Quadrature or imaginary component is plotted along the y-axis.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Wait for Full Window	Check box	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement plots the Inphase component along the x-axis and the Quadrature component along the y-axis. The data is unitless. Only a portion of the measured waveform is displayed. The simulation time scroll bar is used to select the portion to be displayed, while the **Time Span** and **Units** settings determine the number of data points displayed.

If the **Wait for Full Window** check box is selected the measurement will not generate a plot until the full data window can be plotted. If it is not selected then the measurement will generate a plot for as many data points as are available.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

System Waveform: WVFM

Summary

WVFM plots a system simulation time-domain waveform against time.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Wait for Full Window	Check box	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns real values. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

If the **Wait for Full Window** setting is checked, the measurement will not generate a plot if there are fewer than the data window width of symbols available.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

System Histogram: S_HIST

Summary

S_HIST generates a histogram from a system simulation real signal.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Number of Bins	Integer value	>=2
Lower Threshold	Real value	
Upper Threshold	Real value	
Data Window in Samples	Integer value	>=1
Data Window Units	List of options	N/A

Result

This measurement returns histogram values.

Graph Type

This measurement can be displayed on a histogram graph, rectangular graph or tabular grid.

Computational Details

The histogram is generated by sorting the values from the input signal into several bins. The sort criteria depends on whether the **Lower Threshold** setting is less than or greater than the **Upper Threshold** setting.

If **Number of Bins** is 2, a simplified sort is used. An average threshold value is calculated as the average of the **Lower Threshold** and **Upper Threshold** settings. If **Lower Threshold** is less than or equal to **Upper Threshold**, then values less than the average threshold are placed into bin 0 and all other values are placed into bin 1. If **Lower Threshold** is greater than **Upper Threshold**, then values greater than the average threshold are placed into bin 0 and all other values placed into bin 1.

For other values of **Number of Bins**, if the **Lower Threshold** setting is less than the **Upper Threshold** setting, then the first bin contains all values less than **Lower Threshold**. The last bin contains all values greater than or equal to **Upper Threshold**. The boundaries for the intermediate bins are determined from:

$$\frac{(UT-LT)}{N-2}(i-2)+LT \le x_i < \frac{(UT-LT)}{N-2}(i-1)+LT, \qquad i = 2, 3, \ldots, N-1$$

where LT is the **Lower Threshold** setting, UT is the **Upper Threshold** setting, N is the **Number of Bins** setting, and i is the i'th bin (i=1 is the first bin).

If the **Lower Threshold** setting is greater than the **Upper Threshold** setting, then the first bin contains all values greater than **Lower Threshold**. The last bin contains all values less than or equal to **Upper Threshold**. The boundaries for the intermediate bins are determined from:

$$\frac{(UT-LT)}{N-2}(i-2)+LT > x_i \ge \frac{(UT-LT)}{N-2}(i-1)+LT, \qquad i = 2, 3, \ldots, N-1$$

The number of samples sorted is controlled by the **Data Window in Samples** setting. If **Data Window in Samples** is greater than 0, then only a portion of the input waveform is sorted. The simulation time scroll bar is used to control the position of the data window along the input waveform, while the **Data Window in Samples** setting determines how many samples are counted.

If the **Data Window in Samples** setting is 0, then the samples from the first available sample to the simulation time represented by the simulation time scroll bar are sorted. If there are at least **Number of Bins** samples available, a minimum of that many samples are sorted.

For example, if there are 1000 samples available and simulation time scroll bar is set to the left or top-most position, representing T=0, then the first **Number of Bins** samples will be sorted. On the other hand, if the simulation time scroll bar is set to the right or bottom-most position, representing all the available samples, then all 1000 samples will be sorted.

Bit or Symbol Error Rate: BER

Summary

BER generates a Bit Error Rate (BER) or Symbol Error Rate (SER) plot. It works with the BER, BER_EXT, SER and SER_EXT meter blocks.

BER allows you to display the error rate, the actual error counts or the total samples processed. You can also display the statistics at the end of each trial block as well as at the end of each sweep.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Output Type	List of options	N/A
Output Frequency	List of options	N/A

Result

This measurement plots the computed BER or SER along the y-axis and the swept variable (typically E_b/N_0 or E_s/N_0) along the x-axis. When **Output Type** is “Error Rate” the y-axis is typically set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid. A tabular grid is suggested when **Output Frequency** is set to “End of trial blocks”.

BPSK Bit or Symbol Error Rate: BPSK_BERREF

Summary

BPSK_BERREF generates a theoretical BPSK Bit Error Rate reference curve.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Modulation Type	List of options	N/A

Result

The measurement plots a theoretical BPSK bit error probability along the y-axis and the swept variable (typically E_b/N_0) along the x-axis. The y-axis should normally be set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measurement generates a reference curve based on the settings of the meter block selected in the **BER/SER Meter** setting. Values for P_b are calculated for each value specified in the meter's SWPTV parameter.

The following BPSK modulation types are supported:

COHERENT BPSK: [1]

$$P_b = Q\left(\sqrt{\frac{2E_b}{N_0}}\right)$$

where $Q(x)$ is the Gaussian integral or Q-function:

$$Q(x) = \int_{x}^{\infty} \frac{1}{\sqrt{2\pi}} e^{-u^2} du$$

and is approximated numerically, E_b is the average bit energy and N_0 is the noise power spectral density.

OPTIMUM DIFFERENTIAL BPSK: [2]

$$P_b = \frac{1}{2}\exp(-E_b/N_0)$$

SUBOPTIMUM DIFFERENTIAL BPSK: [2]

$$P_b = \frac{1}{2}\exp(-0.8E_b/N_0)$$

where the ideal narrow-band IF filter has bandwidth $W = 0.57/T$, where T is the bit (and symbol) duration.

COHERENT DIFFERENTIALLY ENCODED BPSK: [3]

$$P_b = 2Q\left(\sqrt{\frac{2E_b}{N_0}}\right)\left(1 - Q\left(\sqrt{\frac{2E_b}{N_0}}\right)\right)$$

References

[1] Xiong, F., *Digital Modulation Techniques*, pg. 127
[2] Xiong, F., *Digital Modulation Techniques*, pg. 134
[3] Xiong, F., *Digital Modulation Techniques*, pg. 136

FSK Bit or Symbol Error Rate: FSK_BERREF

Summary

FSK_BERREF generates a theoretical FSK Bit Error Rate or Symbol Error Rate reference curve.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Modulation Type	List of options	N/A
Demodulation Type	List of options	N/A
Statistic Type	List of options	N/A

Result

The measurement plots a theoretical FSK bit or symbol error probability along the y-axis and the swept variable (typically E_b/N_0 or E_s/N_0) along the x-axis. The y-axis should normally be set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measurement generates a reference curve based on the type and settings of the meter block selected in the **BER/SER Meter** setting. If the **Statistic Type** parameter is set to “Auto”, the measurement will compute the bit error probabilities P_b for BER meters and symbol error probabilities P_s for SER meters. Values for P_b or P_s are calculated for each power value specified in the meter’s SWPTV parameter.

The following demodulation types are supported:

COHERENT DEMODULATION:

The curve generated is the upper bound for equal-energy and orthogonal signal sets and coherent detection [1]:

$$P_s \le (M-1)Q\left(\sqrt{\frac{E_s}{N_0}}\right)$$

where $Q(x)$ is the Gaussian integral or Q-function:

$$Q(x) = \int_x^{\infty} \frac{1}{\sqrt{2\pi}} e^{-u^2} du$$

and is approximated numerically, E_s is the average symbol energy, N_0 is the noise power spectral density and M is the number of signal levels as determined by the Modulation Type setting.

NON-COHERENT DEMODULATION:

The curve generated is an approximation of the upper bound for equiprobable, equal-energy, orthogonal MFSK calculated from [2]:

$$P_s \le \frac{M-1}{2} \exp(-E_s/(2N_0))$$

which becomes increasingly accurate as E_s/N_0 increases.

DISCRIMINATOR DEMODULATION:

The curve generated is calculated from [3]:

$$P_s = \left(\frac{1}{2} + \frac{1}{4}\left(\frac{M}{2} + 1\right)\right) \exp(-2E_s/(MN_0))$$

which assumes an IF filter with a sufficiently broad bandwidth and post-detection low pass filter approximated by an ideal integrator.

For some of the probability estimates, the equations may result in a P_s that is greater than 1.0. For these cases, the measurement limits the value of P_s to 1.0.

The measurement estimates the bit error probabilities from the symbol error probabilities using the following approximation [1]:

$$P_b \approx \frac{P_s}{\log_2 M}$$

which is exact if M is a power of 2.

References

[1] Xiong, F., *Digital Modulation Techniques*, pg. 109
[2] Xiong, F., *Digital Modulation Techniques*, pg. 113
[3] Xiong, F., *Digital Modulation Techniques*, pg. 119

PSK BER/SER Reference: PSK_BERREF

Summary

PSK_BERREF generates a theoretical M-Ary PSK Bit Error Rate or Symbol Error Rate reference curve.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Modulation Type	List of options	N/A
Statistic Type	List of options	N/A

Result

The measurement plots a theoretical M-Ary PSK bit or symbol error probability along the y-axis and the swept variable (typically E_b/N_0 or E_s/N_0) along the x-axis. The y-axis should normally be set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measurement generates a reference curve based on the type and settings of the meter block selected in the **BER/SER Meter** setting. If the **Statistic Type** parameter is set to "Auto", the measurement will compute the bit error probabilities P_b for BER meters and symbol error probabilities P_s for SER meters. Values for P_b or P_s are calculated for each power value specified in the meter's SWPTV parameter.

The reference curves generated by the measurement are approximations of the theoretical curves for coherent demodulation [1]. The following equation is used, which is based on $E_s/N_0 \gg 1$:

$$P_s = 2Q\left(\sqrt{\frac{2E_s}{N_0}}\sin\frac{\pi}{M}\right)$$

where $Q(x)$ is the Gaussian integral or Q-function:

$$Q(x) = \int_x^{\infty} \frac{1}{\sqrt{2\pi}} e^{-u^2} du$$

and is approximated numerically, E_s is the average symbol energy, N_0 is the noise power spectral density and M is the number of signal levels.

The measurement computes bit error probabilities from symbol error probabilities using the following approximation [2]:

$$P_b \approx \frac{P_s}{\log_2 M}$$

This approximation assumes Gray coded MPSK, in which case the most likely symbol errors are adjacent symbols that differ only by one bit.

References

[1] Xiong, F., *Digital Modulation Techniques*, pg. 143
[2] Xiong, F., *Digital Modulation Techniques*, pg. 145

QAM Bit or Symbol Error Rate: QAM_BERREF

Summary

QAM_BERREF generates a theoretical QAM Bit Error Rate or Symbol Error Rate reference curve.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Modulation Type	List of options	N/A
Statistic Type	List of options	N/A

Result

The measurement plots a theoretical QAM bit or symbol error probability along the y-axis and the swept variable (typically E_b/N_0 or E_s/N_0) along the x-axis. The y-axis should normally be set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measurement generates a reference curve based on the type and settings of the meter block selected in the **BER/SER Meter** setting. If the **BER/SER Meter** parameter is set to “Auto”, the measurement will compute the bit error probabilities P_b for BER meters and symbol error probabilities P_s for SER meters. Values for P_b or P_s are calculated for each power value specified in the meter’s SWPTV parameter.

When M is an even power of 2 ($M=2^k$, k is even) the following equations are used [1]:

$$P_s = 2P_{\sqrt{M}} - P^2_{\sqrt{M}}$$

$$P_{\sqrt{M}} = \frac{2(\sqrt{2}-1)}{\sqrt{M}} Q\left(\sqrt{\frac{3E_s}{(M-1)N_0}}\right)$$

where $Q(x)$ is the Gaussian integral or Q-function:

$$Q(x) = \int_x^{\infty} \frac{1}{\sqrt{2\pi}} e^{-u^2} du$$

and is approximated numerically, E_s is the average symbol energy, N_0 is the noise power spectral density and M is the number of signal levels.

For other values of M an approximate upper bound is calculated from [1]:

$$P_s \le 1 - \left(1 - 2Q\left(\sqrt{\frac{3E_s}{(M-1)N_0}}\right)\right)^2$$

The measurement computes bit error probabilities from the symbol error probabilities using the following approximation [1]:

$$P_b \approx \frac{P_s}{\log_2 M}$$

This approximation assumes Gray coded square QAM constellations, which is not realizable for all values of M.

References

[1] Xiong, F., *Digital Modulation Techniques*, pp. 438-439

QPSK Bit or Symbol Error Rate: QPSK_BERREF

Summary

QPSK_BERREF generates a theoretical QPSK Bit Error Rate or Symbol Error Rate reference curve.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
BER/SER Meter	System BER/SER Meter	N/A
Modulation Type	List of options	N/A
Statistic Type	List of options	N/A

Result

The measurement plots a theoretical QPSK bit or symbol error probability along the y-axis and the swept variable (typically E_b/N_0 or E_s/N_0) along the x-axis. The y-axis should normally be set to use log scaling.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measurement generates a reference curve based on the type and settings of the meter block selected in the **BER/SER Meter** setting. If the **Statistic Type** parameter is set to Auto, the measurement will compute the bit error probabilities P_b for BER meters and symbol error probabilities P_s for SER meters. Values for P_b or P_s are calculated for each power value specified in the meter's SWPTV parameter.

For QPSK, the symbol error probability is related to the bit error probability by [1]:

$$P_s = 2P_b - P_b^2$$

The following QPSK modulation types are supported:

COHERENT QPSK: [1]

$$P_b = Q\left(\sqrt{\frac{2E_b}{N_0}}\right)$$

where $Q(x)$ is the Gaussian integral or Q-function:

$$Q(x) = \int_x^{\infty} \frac{1}{\sqrt{2\pi}} e^{-u^2} du$$

and is approximated numerically. E_b is the average bit energy, E_s is the average symbol energy and N_0 is the noise power spectral density.

OPTIMUM DIFFERENTIAL QPSK: [2]

The measurement approximates optimum differential QPSK as:

$$P_b \approx Q\left(\sqrt{\frac{4E_b}{N_0}} \sin\frac{\pi}{4\sqrt{2}}\right)$$

SUBOPTIMUM DIFFERENTIAL QPSK: [3]

The measurement approximates suboptimum differential QPSK as:

$$P_b \approx \exp(-0.59 E_b / N_0)$$

COHERENT DIFFERENTIAL QPSK: [4]

The measurement approximates coherent differential QPSK as:

$$P_b \approx 2Q\left(\sqrt{\frac{2E_b}{N_0}}\right)$$

which is applicable at high SNR.

References

[1] Xiong, F., *Digital Modulation Techniques*, pg. 159

[2] Xiong, F., *Digital Modulation Techniques*, pg. 164

[3] Xiong, F., *Digital Modulation Techniques*, pg. 165

[4] Xiong, F., *Digital Modulation Techniques*, pg. 167

Peak Code Domain Error: PCDE

Summary

PCDE is a peak code domain error measurement defined in 3G/CDMA HSDPA specifications [1].

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Output Type	List of options	N/A
Delay Comp.	List of options	N/A
Mag/Phase	List of options	N/A
Standard	List of options	N/A
Measurement Interval	List of options	N/A
Scrambling Code	Integer Value	0 - 8191

Result

This measurement returns real values, either in linear units or in dB.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement must be connected to a VSA meter. The measurement compares the measured signal to the generated test signal from the VSA meter and projects the resulting error signal to each of the channelization codes. PCDE is calculated as the largest value of all projections.

The **Output Type** provides the option of displaying either the peak code domain error (if "Peak CDE" is selected) or code domain errors for each of the channelization codes (if "CDE of each orthogonal code" is selected).

The PCDE measurement also supports basic signal alignment compensation. The **Delay Comp.** setting determines the signal delay compensation performed. When set to "Basic," if either signal has had a signal delay introduced, an appropriate delay will be inserted into both signals to align the signals properly.

The **Mag/Phase** compensation setting determines the adjustments to be made to the magnitude and phase of the measured signal. The "Basic" settings compute a static magnitude scaling and a phase rotation based on the static characteristics of the reference and measured signals. The measured signal is then adjusted by this scaling and/or rotation prior to computation of the error vector.

The **Standard** selection provides a placeholder for communication standards that could be added in the future. Currently, only 3G CDMA/HSDPA is supported.

The **Measurement Interval** determines whether the measurement is performed over the whole frame or a single time slot. A specific time slot (0 - 15) can be specified.

The **Scrambling Code** window allows the user to specify the scrambling code used at the transmitter. Available codes range from 0 to 8191.

The PCDE measurement is performed in the following steps:

1. Calculate the error signal as the difference between the measured (M) and the reference (R) signals, which are fed to the inputs of the VSA. For each measurement, NS vectors e of length SF are generated, where NS is the number of symbols in the measurement interval and SF is the number of chips per symbol.
2. To achieve meaningful results it is necessary to descramble e, leading to e'.
3. Take the orthogonal vectors of the channelization code set C (all codes belonging to one spreading factor) as defined in [2] (range +1, -1) and normalize by the norm of the vectors to produce Cnorm= C/sqrt(SF).
4. Calculate the inner product of e' with Cnorm. Do this for all symbols of the measurement interval and for all codes in the code space. This gives an array of format K x NS, each value representing an error-vector representing a specific

symbol and a specific code, which can be exploited in a variety of ways. (K: total number of codes in the code space.)

5. Calculate K RMS values, each RMS value unifying NS symbols within one code. (These values can be called “AbsoluteCodeEVMs” [Volt].)

6. Find the peak value among the K “AbsoluteCodeEVMs.” (This value can be called “AbsolutePeakCodeEVM” [Volt].)

7. Calculate PCDE according to:

$$PCDE = 10 \cdot \log \frac{AbsolutePeakCodeEVM^2}{RMS(R)^2}$$

where PCDE is a relative value in dB.

References

[1] ETSI TS 125 141 V6.11.0 (2005-09), Section E 2.6.2.

[2] ETSI TS 125 213 V6.3.0 (2005-06)

Eye Diagram: EYE

Summary

EYE generates an eye diagram of a system simulation time-domain waveform.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns unitless real values. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement displays a waveform using overlapping traces, similar to an oscilloscope. The width of the display is determined by the **Time Span** value (width) and units settings. The time axis displays the time starting with the first sample displayed.

The total number of symbols displayed is the **Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings. This measurement will not display any data until this number of symbols worth of data is available.

The **Start Offset** setting controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Amplitude: EYE_AMPLITUDE

Summary

EYE_AMPLITUDE computes the amplitude metric of an eye diagram. The amplitude metric is defined as:

$$Amplitude = Level\_One\_mean - Level\_Zero\_mean$$

Level_One_mean and *Level_Zero_mean* are the mean Y values of Level One and Level Zero, respectively. The computation of these values is performed in a manner similar to the Eye Level Info measurement *EYE_LEVEL*.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the “*Simulation Basics*” chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

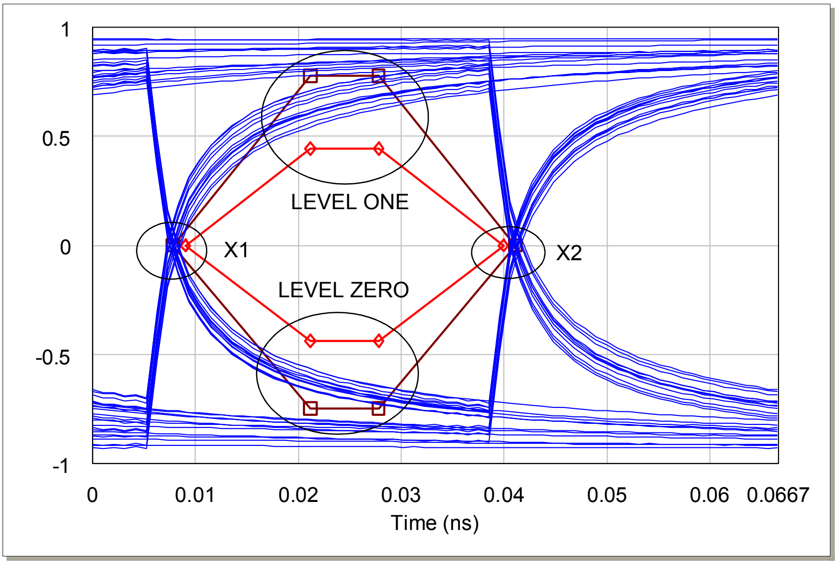
The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Corners: EYE_CORNERS

Summary

EYE_CORNERS plots the two eye crossings (X1 and X2), the Level One edges, and the Level Zero edges of an eye diagram. This measurement is often used with the Eye Diagram measurement ***EYE*** to visualize the corners of the eye:



Six points are plotted: X1, the left edge of Level One, the right edge of Level One, X2, the right edge of Level Zero, and the left edge of Level Zero. The left and right edges of Level One and Level Zero are the edges of the window used to compute those values.

The x values of X1 and X2 can be the mean, the mean +/- the standard deviation, or the mean +/- 3 times the standard deviation. The **Crossing Display** setting determines which value displays.

Similarly, the y values of Level One and Level Zero can be the mean, the mean +/- the standard deviation, or the mean +/- 3 times the standard deviation. The **Level Display** setting determines which value displays.

In the previous graph, the brown curve plots the six points using only the mean, while the red curve plots the six points using the mean +/- 3 times the standard deviation.

EYE_CORNERS uses the same settings and algorithms as the Eye Crossing Info measurement *EYE_CROSSING* and the Eye Level Info measurement *EYE_LEVEL* for determining the crossing and level information. See those measurements for details.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%
Eye Window Center (%)	Percent	10.5% to 89.5%, default 50%
Crossing Display	List of Options	N/A
Level Display	List of Options	N/A
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values. The x-axis is scrollable simulation time.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Crossing Info: EYE_CROSSING

Summary

EYE_CROSSING locates the left (X1) or right (X2) crossing point of an eye diagram and displays information about that crossing point. The following information is available for each crossing point:

- Y Value at the crossing
- Mean
- Sigma (Standard Deviation)
- Lower and Upper Peaks
- Number of traces at the crossing

The **Output Type** determines which value displays.

The *EYE_CORNERS* measurement along with the Eye Diagram measurement *EYE* can be used to visualize the eye crossings on an eye diagram.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Output Type	List of Options	N/A
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. The parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value whose units depend upon the **Output Type**. For "Y Crossing" the value is unitless. For the mean, sigma and peak options the units are time. For the count options the value is unitless.

The measurement is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Computational Details

EYE_CROSSING uses a peak detection algorithm to locate the crossing points. At a given Y level, the algorithm generates a histogram across the time axis. The histogram measures the number of times the traces cross the Y level at each binned time value. The time coordinate at which a trace crosses the specified Y level is linearly interpolated from the trace samples.

The histogram is then smoothed by averaging adjacent bins. The number of bins used in the averaging is determined by the **Peak Smoothing** setting, which specifies the percentage of the eye diagram span over which to average bins.

After smoothing peaks are located by applying a threshold to the averaged counts. A peak consists of a contiguous set of bins whose counts exceed the threshold level.

The threshold is determined by the **Peak Threshold** setting, which is specified as a percentage of the full Y axis range.

Once the peaks are found, their edges are extended to meet the edges of the adjacent peaks at the mid-point between the edges:

$$Edge_{New} = (UpperEdge_{i-1} + LowerEdge_i)/2$$

where the upper edge of the previous peak and the lower edge of the following peak are set to $Edge_{New}$.

The mean and variance are computed for each peak using the counts within the boundary edges of the peak.

The strongest peak is then chosen as one crossing point, with the stronger of the two adjacent peaks chosen as the other crossing point.

By default, the Y level is determined automatically. The **Y Crossing Level** setting lets you explicitly specify the Y level. When determined automatically, several different Y levels are selected and peaks found. One of the Y levels is then chosen, with stronger weight given to Y levels where only two to three peaks are found.

Because of this weighting the **Time Span** should normally be set so the eye diagram contains two to three crossing points.

Eye Extinction Ratio: EYE_EXTRATIO

Summary

EYE_EXTRATIO computes an extinction ratio or percentage for an eye diagram. Several different outputs are supported:

Power Ratio:

$$Ratio = \frac{Level\_One\_mean^2}{Level\_Zero\_mean^2}$$

Power %:

$$Percent = \frac{Level\_Zero\_mean^2}{Level\_One\_mean^2} \cdot 100$$

Voltage Ratio:

$$Ratio = \frac{Level\_One\_mean - Min(Y)}{Level\_Zero\_mean - Min(Y)}$$

Voltage %:

$$Percent = \frac{Level\_Zero\_mean - Min(Y)}{Level\_One\_mean - Min(Y)} \cdot 100$$

Level_One_mean and *Level_Zero_mean* are the mean Y values of Level One and Level Zero, respectively. *Min*(*Y*) is the minimum Y value of the signal. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

The **Output Type** determines which equation is used.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0

Name	Type	Range
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Fall Time: EYE_FALLTIME

Summary

EYE_FALLTIME computes the fall time for an eye diagram, which is the average time required to transition from Level One to Level Zero. The fall time is computed as the difference between the mean time at which traces transitioning from Level One to Level Zero cross one y-axis level and the mean time at which the traces cross another y-axis level.

By default the start of the transition is the y-axis level that is 80% of the distance between Level Zero and Level One, measured from Level Zero. Similarly, the end of the transition is the y-axis level that is 20% of the distance. These defaults can be changed using the **Level A Offset** and **Level B Offset** secondary settings.

The computation of the edges of the transition is performed in a manner similar to the *EYE_TRANSITION* measurement.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
*Eye Window Width (%)	Percent	1% to 80%, default 20%
*Level A Offset (%)	Percent	0% to 100%, default 20%
*Level B Offset (%)	Percent	0% to 100%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value with units of time and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Height: EYE_HEIGHT

Summary

EYE_HEIGHT computes the height metric of an eye diagram. The height metric is defined as:

$$Height = (Level\_One\_mean - 3 \cdot Level\_One\_sigma) - (Level\_Zero\_mean + 3 \cdot Level\_Zero\_sigma)$$

Level_One_mean and *Level_Zero_mean* are the mean Y values of Level One and Level Zero, respectively. *Level_One_sigma* and *Level_Zero_sigma* are the standard deviations of the Y values of Level One and Level Zero, respectively. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Inverse Extinction Ratio: EYE_INVEXTRATIO

Summary

EYE_INVEXTRATIO computes an extinction ratio or percentage for an eye diagram. Several different outputs are supported:

Power Ratio:

$$Ratio = \frac{Level\_Zero\_mean^2}{Level\_One\_mean^2}$$

Power %:

$$Percent = \frac{Level\_One\_mean^2}{Level\_Zero\_mean^2} \cdot 100$$

Voltage Ratio:

$$Ratio = \frac{Level\_Zero\_mean - Max(Y)}{Level\_One\_mean - Max(Y)}$$

Voltage %:

$$Percent = \frac{Level\_One\_mean - Max(Y)}{Level\_Zero\_mean - Max(Y)} \cdot 100$$

Level_One_mean and *Level_Zero_mean* are the mean Y values of Level One and Level Zero, respectively. $Min(Y)$ is the minimum Y value of the signal. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

The **Output Type** determines which equation is used.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0

Name	Type	Range
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Jitter: EYE_JITTER

Summary

EYE_JITTER computes the jitter metric of an eye diagram. Two metrics are available:

Peak-Peak:

$$Jitter_{PkPk} = max(X1\_upperPk - X1\_lowerPk, X2\_upperPk - X2\_lowerPk)$$

RMS:

$$Jitter_{rms} = max(X1\_sigma, X1\_sigma)$$

X1_upperPk and *X2_upperPk* are the maximum X values of the X1 and X2 crossing points, respectively. *X1_lowerPk* and *X2_lowerPk* are the minimum X values of the X1 and X2 crossing points, respectively.

X1_sigma and *X2_sigma* are the standard deviations of the X values of the X1 and X2 crossing points, respectively.

The computation of these values is performed in a manner similar to the Eye Crossing Info measurement *EYE_CROSSING*.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Output Type	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value with units of time and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Level Info: EYE_LEVEL

Summary

EYE_LEVEL displays information about the Level One or Level Zero points of an eye diagram. The following information is available:

- Level One and Level Zero Mean
- Level One and Level Zero Sigma (Standard Deviation)
- Level One and Level Zero Lower and Upper Peaks
- Number of points used to compute Level One or Level Zero statistics.
- Maximum and Minimum Y values in the entire eye diagram.

The **Output Type** determines which value displays.

Level One represents the vertical amplitude at the top of the signal (more positive) while Level Zero represents the vertical amplitude at the bottom of the signal (less positive). Both levels are measured statistically within an eye window, which is specified through the **Eye Window Width** and **Eye Window Center** settings.

The center of the window is normally set to 50%. For NRZ signals the width of the window is typically 20%. For RZ signals the width of the window is typically 5%.

The *EYE_CORNERS* and *EYE* measurements can be used to visualize the eye crossings on an eye diagram.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%
Eye Window Center (%)	Percent	10.5% to 89.5%, default 50%
Output Type	List of Options	N/A

Name	Type	Range
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

The **Start Offset** setting controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Computational Details

EYE_LEVEL first divides the eye diagram horizontally into an upper section and a lower section at the Y level of the crossings. The Y level can be set manually through the **Y Crossing Level** setting, or determined automatically similar to the Eye Crossing Info measurement *EYE_CROSSING*.

The crossing points are also used to determine the left and right edges of the portion of the eye diagram within which the level information is measured. The **Eye Window**

Width and **Eye Window Center** settings determine these edges. These settings are specified as a percentage of the distance between the mean time values of the two crossing points.

The statistics for each level are computed over a set of time values within the eye window. At each time value the Y level for each trace is linearly interpolated from the trace samples. If the Y level is less than the Y crossing level the point is added to the Level Zero statistics, otherwise it is added to the Level One statistics.

The time values are set to start at the left edge and end on the right edge, inclusive, with a step of approximately 1% of the distance between the two crossing points.

Eye Overshoot: EYE_OVERSHOOT

Summary

EYE_OVERSHOOT computes the overshoot of Level One (the upper level) of an eye diagram. The overshoot is defined as:

$$Overshoot = Max(Y) - Level\_One\_mean$$

$Max(Y)$ is the maximum Y value of the signal within the eye diagram. *Level_One_mean* is the mean Y value of Level One. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Q Factor: EYE_QFACTOR

Summary

EYE_QFACTOR computes a Q factor for an eye diagram. The Q factor is computed as:

$$QFactor = \frac{Level\_One\_mean - Level\_Zero\_mean}{Level\_One\_sigma + Level\_Zero\_sigma}$$

Level_One_mean and *Level_Zero_mean* are the mean Y values of Level One and Level Zero, respectively, while *Level_One_sigma* and *Level_Zero_sigma* are the standard deviations. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Rise Time: EYE_RISETIME

Summary

EYE_RISETIME computes the rise time for an eye diagram, which is the average time required to transition from Level Zero to Level One. The rise time is computed as the difference between the mean time at which traces transitioning from Level Zero to Level One cross one y-axis level and the mean time at which the traces cross another y-axis level.

By default the start of the transition is the y-axis level that is 20% of the distance between Level Zero and Level One, measured from Level Zero. Similarly, the end of the transition is the y-axis level that is 80% of the distance. These defaults can be changed using the **Level A Offset** and **Level B Offset** secondary settings.

The computation of the edges of the transition is performed in a manner similar to the *EYE_TRANSITION* measurement.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
*Eye Window Width (%)	Percent	1% to 80%, default 20%
*Level A Offset (%)	Percent	0% to 100%, default 20%
*Level B Offset (%)	Percent	0% to 100%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value with units of time and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

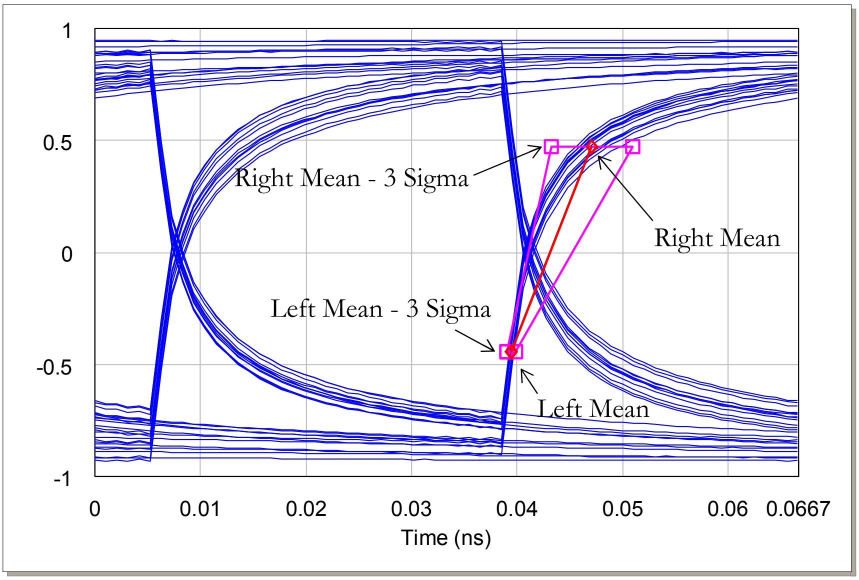
Eye Transition: EYE_TRANSITION

Summary

EYE_TRANSITION displays information about the transitions between Level One and Level Zero of an eye diagram. Transitions are quantified by first identifying all traces that pass from Level One to Level Zero (falling transition) or from Level Zero to Level One (rising transition), depending upon the desired transition. The start and end times of the transition are the means of the x-axis values (time) at which the traces cross specific y-axis levels. The start time corresponds to the Left side of the transition, while the end time corresponds to the Right side of the transition.

By default, the y-axis level for the Left side is 20% of the distance between Level Zero and Level One for rising transitions and 80% for falling transitions. Similarly, the y-axis level for the Right side is 80% for rising transitions and 20% for falling transitions. These settings may be changed using the secondary **Level Offset A** and **Level Offset B** settings.

This measurement can be used to present an overlay of the transition on an eye diagram graph:



In this graph, the red solid curve is the EYE_TRANSITION measurement configured to display the Left and Right Means, while the pink dashed curve is the measurement configured to display the means +/- 3 sigma.

The measurement can also display the following values individually:

- Left or Right Mean
- Left or Right Sigma (Standard Deviation)
- Left or Right Lower Peak
- Left or Right Upper Peak
- Number of traces in the left or right edge

The **Output Type** determines what is displayed.

EYE_TRANSITION uses many of the same settings and algorithms as the Eye Crossing Info measurement *EYE_CROSSING* and the Eye Level Info measurement *EYE_LEVEL* for determining the crossing and level information. See those measurements for details.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A
Transition Type	List of Options	N/A
Output Type	List of Options	N/A
*Eye Window Width (%)	Percent	1% to 80%, default 20%
*Eye Window Center (%)	Percent	10.5% to 89.5%, default 50%
*Level A Offset (%)	Percent	0% to 100%, default 20%
*Level B Offset (%)	Percent	0% to 100%, default 20%

Name	Type	Range
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value whose units depends upon the **Output Type**. For the mean, sigma and peak options the units are time. For the count options the value is unitless. When displaying the transition bands, the x-axis units are time while the y-axis units are unitless.

This measurement is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Undershoot: EYE_UNDERSHOOT

Summary

EYE_UNDERSHOOT computes the undershoot of Level Zero (the lower level) of an eye diagram. The undershoot is defined as:

$$Undershoot = Level\_Zero\_mean - Min(Y)$$

Level_Zero_mean is the mean Y value of Level Zero. *Min*(*Y*) is the minimum Y value of the signal within the eye diagram. The computation of these values is performed in a manner similar to the Eye Level Info measurement EYE_LEVEL.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns unitless real values and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

Eye Width: EYE_WIDTH

Summary

EYE_WIDTH computes the width metric of an eye diagram. The width metric is defined as:

$$Width = (X2\_mean - 3 \cdot X2\_sigma) - (X1\_mean + 3 \cdot X1\_sigma)$$

$X1\_mean$ and $X2\_mean$ are the mean X values of the two crossing points. $X1\_sigma$ and $X2\_sigma$ are the standard deviations of the X values of the two crossing points. The computation of these values is performed in a manner similar to the Eye Crossing Info measurement EYE_CROSSING.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Maximum Number of Traces	Integer value	>0
Start Offset	Real value	>=0
Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value with units of time and is scrollable in simulation time.

Graph Type

This measurement can be displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

The **Time Span** should normally be set so it contains two to three eye crossings (typically two to three symbol periods). The eye crossing detection algorithm works best with this number of crossings.

The maximum number of symbols included in the eye diagram is the **Maximum Number of Traces** value times the time span in symbols, as determined by the **Time Span** and **Units** settings.

Start Offset controls the alignment of the eye diagram openings relative to the x-axis. As the offset is increased, the eye openings shift to the left.

EDGE EVM: EDGE_EVM

Summary

EDGE_EVM computes the error vector magnitude (EVM) measurements used in the GSM/EDGE EVM modulation accuracy specifications in [1].

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM_VSA meter	N/A
Measurement Type	List of options	N/A
X-axis Display	List of options	N/A
Display cumulative average	Check box	N/A
Display overall maximum	Check box	N/A
Display reference limit, normal	Check box	N/A
Display reference limit, extreme	Check box	N/A

Result

This measurement returns real values. If more than one display option is checked, multiple traces are displayed, with the first trace corresponding to the top-most checked display option.

The X-axis display parameter determines what is displayed for the x-axis. For the power options, the x-axis values are in the log power units (dBm or dB) specified in the project options.

The Y-axis units are percent (i.e. 10 = 10%).

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement must be connected to a GSM_VSA meter. The measurement compares the measured signal to the generated test signal from the VSA meter. This is performed in several steps:

1. At the start of each simulation sweep, a complex scaling factor is computed based on the first burst detected. This scaling factor is used to compensate for C1 and W in [2], and is computed from:

$$s = \frac{\sum_{n \in N} M^*(n) \cdot R(n)}{\sum_{n \in N} |M(n)|^2}$$

   where M is the measured signal, M^* is the complex conjugate of the measured signal, R is the reference signal, and N is the set of samples in the burst.

2. The measured signal is then scaled by s, passed through the measurement filter described in [1], and sampled at the start of each symbol. The reference signal is also passed through the measurement filter and sampled at the start of each symbol to obtain S(k).
3. The error vector E(k) is then calculated by subtracting S(k) from the filtered and scaled measured signal. The carrier feedthrough offset C0 is ignored by this measurement because the carrier is assumed to be ideal.
4. The metric described by the Measurement Type parameter is then computed and plotted on the y-axis.

RMS EVM:

$$y_{Percent} = \sqrt{\frac{\sum_{k \in K} |E(k)|^2}{\sum_{k \in K} |S(k)|^2}} \cdot 100$$

An RMS EVM value is computed for each time slot.

PEAK EVM:

The peak EVM value is the maximum symbol error vector magnitude value in a burst, and is computed from:

$$y_{Percent} = max\left(\sqrt{\frac{|E(k)|^2}{\frac{1}{N}\sum_{k \in K}|S(k)|^2}}\right)$$

where N is the number of symbols in the burst. Peak EVM values are computed and output only every 8'th time slot, starting with the first time slot.

95TH PERCENTILE:

The 95th percentile EVM is computed by first calculating the symbol error vector magnitude for the burst:

$$EVM(k) = \sqrt{\frac{|E(k)|^2}{\frac{1}{N}\sum_{k \in K}|S(k)|^2}}$$

where N is the number of symbols in the burst. The set of EVM values is then sorted in ascending order, and the EVM value that is greater than 95% of the EVM values is output. This value is:

$$y_{Percent} = EVM(k_{95th}) \cdot 100$$

were k_{95th} is the 136th EVM smallest value.

A 95th percentile EVM measurement is computed for each time slot.

OTHER DETAILS

This measurement uses the training sequence bits to detect the start of each burst.

If the X-axis is set to display power, the power displayed is cumulative average of the instantaneous power of the reference or measured signal up to and including the time slot being measured.

References

[1] 3GPP TS 45.005 V5.2.0 (2001-11), Section 4.6.2.

[2] 3GPP TS 45.005 V5.2.0 (2001-11), Annex G.

GSM Adjacent Channel Power: GSM_ACP

Summary

GSM_APC computes several of the adjacent channel power (ACP) measurements described in [1]. The ACP measurements verify the output RF spectrum due to the modulation and the switching transients [2].

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM VSA Meter	N/A
Measurement type	List of options	N/A
Time slot to measure	List of options	N/A
Display measured dBc	Check box	N/A
Display reference mask (dBc)	Check box	N/A

Result

This measurement returns real values. If you select more than one display option, multiple traces are displayed, with the first trace corresponding to the top-most selected display option.

The x-axis values are frequency values. The y-axis values are dBc values.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement uses the training sequence bits of the measured signal to detect the start of the burst in each time slot. If a burst cannot be detected for the current time slot, an estimate of the start of the time slot is made based on the last detected burst and the duration of a time slot.

The ACP measurements are computed at different channel offsets, which are determined from the measurement type and the specifications [1]. The Measurement type parameter determines the specific measurement to be performed.

For each channel offset, including the carrier frequency, the measured signal is passed through an FIR filter to obtain the signal at that offset. Each FIR filter is configured to have the same number of coefficients as samples in each time slot, and to have a frequency response of 0 dB over the resolution bandwidth for the filter (this is determined by the channel offset and measurement type).

For the time slots indicated by the Time slot to measure parameter, the ACP measurement is then made.

DUE TO MODULATION:

The time gated cumulative average power is computed from the filtered signal for each channel starting at the end of the training/synchronization portion of the burst or the 50% point of the useful part of the burst, whichever is later, and continuing up to the 90% point of the useful part of the burst. (Compensation is made for the FIR filter delay).

The cumulative average power is computed for the carrier frequency using the same time gating.

The y-axis values for each frequency offset are then computed by converting the cumulative average power for the channel to dB and subtracting the cumulative average power of the carrier frequency.

DUE TO SWITCHING TRANSIENTS:

The overall peak instantaneous power for the time slot is measured.

The cumulative average power is also computed for the carrier frequency, using a 300 kHz bandwidth filter.

The y-axis value for each frequency offset is then measured by converting the peak instantaneous power to dB and subtracting the carrier frequency cumulative average power in dB.

OTHER DETAILS:

The reference curves displayed if the Display reference mask option is checked are based on the modulation of the time slot selected by the Time slot to measure parameter. If Time slot to measure is set to “All” time slot 0 is used.

The reference curves are based on the tables found in [2], and assume the maximum power level and normal operation.

References

[1] 3GPP TS 51.021 V4.0, Section 6.5.
[2] 3GPP TS 45.005 V5.2.0 (2001-11), Section 4.2.

GSM GMSK Modulation Accuracy, Frequency Error: GSM_FRQ

Summary

GSM_FRQ performs the frequency error measurement described in [1, 2]. The frequency error measurement is used to verify GMSK modulation accuracy.

The GSM_PHS measurement can be used to perform the phase error measurement that is also part of the GMSK modulation accuracy measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM VSA Meter	N/A
X-Axis display	List of options	N/A
Y-Axis display	List of options	N/A
Display overall maximum	Check box	N/A
Display cumulative average	Check box	N/A
Display reference limit	Check box	N/A

Result

This measurement returns real values. If more than one display option is checked, multiple traces are displayed, with the first trace corresponding to the top-most checked display option.

The X-axis display parameter determines what is displayed for the x-axis. For the power options, the x-axis values are in the log power units (dBm or dB) specified in the project options.

The Y-axis units are determined by the Y-Axis display parameter. If Y-Axis display is set to "ppm" and the measured signal has a carrier frequency of 0, the results are displayed in Hz.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement must be connected to a GSM_VSA meter. The measurement compares the measured signal to the generated test signal from the VSA meter in several steps:

1. The phase trajectory (phase versus time) over the useful part of each burst is calculated for both the reference and the measured signal. The useful part of the burst starts halfway through the first symbol and continues for 147 symbols. The phase trajectory is calculated using the following:

$$\begin{aligned}
\phi[i] &= \phi[i-1] + \Delta\theta[i] + adj[i] \\
\phi[0] &= \theta[0] \\
\Delta\theta[i] &= \theta[i] - \theta[i-1] \\
adj[i] &= -2\pi \qquad \text{if } \Delta\theta[i] > \pi \\
adj[i] &= 2\pi \qquad \text{if } \Delta\theta[i] \le -\pi \\
adj[i] &= 0 \qquad \text{otherwise}
\end{aligned}$$

   The $adj[i]$ term is used to compensate for +/- π wraparound.

2. The phase difference trajectory is calculated by subtracting the phase trajectory of the reference signal from the phase trajectory of the measured signal:

$$\begin{aligned}
\phi_{Diff}[i] &= \phi_{Meas}[i] - \phi_{Ref}[i] + \phi_{Adj}[i] \\
\phi_{Adj}[i] &= -2\pi \qquad \text{if } (\phi_{Meas}[i] - \phi_{Ref}[i]) > \pi \\
\phi_{Adj}[i] &= 2\pi \qquad \text{if } (\phi_{Meas}[i] - \phi_{Ref}[i]) \le -\pi \\
\phi_{Adj}[i] &= 0 \qquad \text{otherwise}
\end{aligned}$$

   The $\phi_{Adj}[i]$ term is used to compensate for +/- π wraparound.

3. The linear regression line for the phase difference trajectory is then calculated using least squares:

$$\phi_{Regr}[i] = m \cdot i + b$$

$$m = \frac{\sum (i \cdot \phi_{Diff}[i]) - \dfrac{(\sum i) \cdot (\sum \phi_{Diff}[i])}{N}}{\sum i^2 - \dfrac{(\sum i)^2}{N}}$$

$$b = \frac{\sum \phi_{Diff}[i] - m \sum i}{N}$$

4. The frequency error in Hertz is then calculated from the slope of the linear regression line for the phase difference trajectory:

$$f_{Err} = \frac{m}{2\pi} \cdot f_s$$

where f_s is the sampling frequency of the signal. The frequency error in ppm is calculated using the carrier frequency for the measured signal:

$$ppm_{Err} = \frac{f_{Err}}{f_{Carrier}} \cdot 1e6$$

OTHER DETAILS

This measurement uses the training sequence bits to detect the start of each burst.

If the X-axis is set to display power, the power displayed is cumulative average of the instantaneous power of the reference or measured signal up to and including the time slot being measured.

References

[1] 3GPP TS 51.021 V4.0.0 (2001-11), Section 6.2.2.
[2] 3GPP TS 51.010-1 V4.6.0 (2001-12), Section 13.1.

GSM GMSK Modulation Accuracy, Phase Error: GSM_PHS

Summary

GMS_PHS performs the phase error measurement described in [1, 2, 3]. The phase error measurement is used to verify GMSK modulation accuracy.

The GSM_FRQ measurement can be used to perform the frequency error measurement that is also part of the GMSK modulation accuracy measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM VSA Meter	N/A
Measurement type	List of options	N/A
X-Axis display	List of options	N/A
Display overall maximum	Check box	N/A
Display cumulative average	Check box	N/A
Display reference limit	Check box	N/A

Result

This measurement returns real values. If more than one display option is checked, multiple traces are displayed, with the first trace corresponding to the top-most checked display option.

The X-axis display parameter determines what is displayed for the x-axis. For the power options, the x-axis values are in the log power units (dBm or dB) specified in the project options.

The Y-axis units are angular.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement must be connected to a GSM_VSA meter. The measurement compares the measured signal to the generated test signal from the VSA meter. This is performed in several steps:

1. The phase trajectory (phase versus time) over the useful part of each burst is calculated for both the reference and the measured signal. The useful part of the burst starts halfway through the first symbol and continues for 147 symbols. The phase trajectory is calculated using the following:

$$\phi[i] = \phi[i-1] + \Delta\theta[i] + adj[i]$$
$$\phi[0] = \theta[0]$$
$$\Delta\theta[i] = \theta[i] - \theta[i-1]$$
$$adj[i] = -2\pi \qquad \text{if } \Delta\theta[i] > \pi$$
$$adj[i] = 2\pi \qquad \text{if } \Delta\theta[i] \le -\pi$$
$$adj[i] = 0 \qquad \text{otherwise}$$

   The $adj[i]$ term is used to compensate for +/- π wraparound.

2. The phase difference trajectory is calculated by subtracting the phase trajectory of the reference signal from the phase trajectory of the measured signal:

$$\phi_{Diff}[i] = \phi_{Meas}[i] - \phi_{Ref}[i] + \phi_{Adj}[i]$$
$$\phi_{Adj}[i] = -2\pi \qquad \text{if } (\phi_{Meas}[i] - \phi_{Ref}[i]) > \pi$$
$$\phi_{Adj}[i] = 2\pi \qquad \text{if } (\phi_{Meas}[i] - \phi_{Ref}[i]) \le -\pi$$
$$\phi_{Adj}[i] = 0 \qquad \text{otherwise}$$

   The $\phi_{Adj}[i]$ term is used to compensate for +/- π wraparound.

3. The linear regression line for the phase difference trajectory is then calculated using least squares:

$$\phi_{Regr}[i] = m \cdot i + b$$

$$m = \frac{\sum(i \cdot \phi_{Diff}[i]) - \frac{(\sum i) \cdot (\sum \phi_{Diff}[i])}{N}}{\sum i^2 - \frac{(\sum i)^2}{N}}$$

$$b = \frac{\sum \phi_{Diff}[i] - m \sum i}{N}$$

4. The phase error is calculated by subtracting the phase difference linear regression line from the phase difference trajectory:

$$\phi_{Err}[i] = \phi_{Diff}[i] - \phi_{Regr}[i]$$

5. The metric described by the Measurement Type parameter is then computed and plotted on the y-axis.

RMS PHASE ERROR:

$$y_{Rms} = \sqrt{\frac{\sum(\phi_{Err}[i])^2}{N}}$$

An RMS value is computed for each time slot.

PEAK PHASE ERROR:

The peak value is simply the maximum absolute phase error value.

OTHER DETAILS

This measurement uses the training sequence bits to detect the start of each burst.

If the X-axis is set to display power, the power displayed is cumulative average of the instantaneous power of the reference or measured signal up to and including the time slot being measured.

References

[1] 3GPP TS 45.005 V5.2.0 (2001-11), Section 4.6.1.
[2] 3GPP TS 51.010-1 V4.6.0 (2001-12), Section 13.1.
[3] 3GPP TS 51.021 V4.0.0 (2001-11), Section 6.2.2.

GSM Power vs. Time: GSM_PVT

Summary

GMS_PVT displays power versus time for GSM bursts. This measurement is useful in determining if the GSM-EDGE "Output Level Dynamic Operation" specifications in [1] are satisfied.

Use either the GSM_VSA or GSM_TSIG block to generate the GSM test signal.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM VSA Meter	N/A
View type	List of options	N/A
Time slot to measure	Integer value	>=0, <=7
Display average	Check box	N/A
Display maximum levels	Check box	N/A
Display minimum levels	Check box	N/A
Display upper reference mask	Check box	N/A
Display lower reference mask	Check box	N/A

Result

This measurement returns real values. If more than one display option is checked, multiple traces are displayed, with the first trace corresponding to the top-most checked display option.

The y-axis values are in dB and are relative to the power level specified in the GSM_VSA or GSM_TSIG block generating the test signal.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement uses the training sequence bits of the measured signal to detect the start of the burst in each time slot. If a burst cannot be detected for the current time slot, an estimate of the start of the time slot is made based on the last detected burst.

The instantaneous power of each sample in the burst is measured and a cumulative average for each sample index in each time slot is kept. The overall minimum and maximum are also kept for each sample index. These values are displayed if the appropriate checkbox is checked.

If multiple time slots are selected for display, the first time slot displayed is determined from both the **Time slot to display** setting and the number of time slots to be displayed. If the **Time slot to display** setting plus the number of time slots is less than or equal to 8, the **Time slot to display** setting indicates the first time slot displayed. If the sum is greater, the first time slot displayed will be 8 minus the number of time slots to display.

References

[1] 3GPP TS 45.005 V5.2.0 (2001-11), Section 4.5.

[2] 3GPP TS 45.005 V5.2.0 (2001-11), Annex B.

GSM Output RF Spectrum: GSM_SPEC

Summary

GSM_SPEC displays a power spectrum loosely compatible with the GSM-EDGE "Output RF Spectrum" specifications in [1]. It can also display the spectrum masks found in [2].

NOTE. This measurement does NOT perform time gating on the signal as required by the specifications. The GSM_ACP measurement does perform the required time gating.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
GSM Vector Signal Analyzer	GSM VSA Meter	N/A
Time slot to measure	List of options	N/A
Display signal spectrum	Check box	N/A
Display reference mask	Check box	N/A
Frequency axis relative to center frequency	Check box	N/A

Result

This measurement returns real values. If you select more than one display option, multiple traces display, with the first trace corresponding to the top-most selected display option.

The y-axis values are dBc values.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement uses the training sequence bits of the measured signal to detect the start of the burst in each time slot. If a burst cannot be detected for the current time slot, an estimate of the start of the time slot is made based on the last detected burst and the duration of a time slot.

The power spectrum is computed using a periodogram with cumulative averaging, 0% overlap and a Hanning window. The width of the FFT used to perform the periodogram is set to the number of samples in each time slot (rounded down). Samples from time slots not indicated by the Time slot to measure parameter are ignored.

The set of frequency values to be displayed is based on the following:

For frequencies between -1800 kHz and 1800 kHz offset, the frequencies at 30 kHz increments are displayed. This includes the center frequency (0 kHz offset). Above 1800 kHz and below -1800 kHz frequency values are computed at 100 kHz increments up to +/- $f_s/2$.

For each frequency to be displayed, a power estimate is made by integrating the power spectrum from the frequencies halfway between the next lower frequency value and the next higher frequency value.

A similar power estimate is made for the center frequency using a bandwidth of 30 kHz and a bandwidth of 100 kHz. If the power estimate for the center frequency is near 0, the center frequency power is assumed to be 0 dB.

The y-axis values are then computed by dividing the power estimate for each frequency value by the power estimate for the center frequency using the same bandwidth (i.e. frequencies below 1800 kHz use the 30 kHz bandwidth, all other frequencies use the 100 kHz bandwidth).

The reference curves displayed if the **Display reference mask** option is checked are based on the modulation of the time slot selected by the **Time slot to measure** parameter. If **Time slot to measure** is set to “All” time slot 0 is used.

The reference curves are based on the tables found in [1], and assume the maximum power level and normal operation.

References

[1] 3GPP TS 45.005 V5.2.0 (2001-11), Section 4.2.
[2] 3GPP TS 45.005 V5.2.0 (2001-11), Annex A.
[3] 3GPP TS 51.021 V4.0, Section 6.5.1.

Swept Analog Carrier to Interference+Noise Ratio C/(I+N): CINR_A

Summary

CINR_A computes an estimate of the carrier to interference plus noise ratio C/(I+N). The measurement utilizes two measured signals. One signal is the output of the device under test (DUT) with the desired input signal on, which is carrier+interference+noise. The other signal is the output of the DUT with the desired input signal off, which is interference+noise.

The power of the two signals over a specified frequency band is computed. C/(I+N) is then approximated as:

$$\frac{C}{N+I} = \frac{P_{SigOn} - P_{SigOff}}{P_{SigOff}}$$

where P_{SigOn} is the total power in the frequency band of the signal with the desired input signal on and P_{SigOff} is the total power in the frequency band of the signal with the desired input signal off.

NOTE. This method of measuring C/(I+N) is only an approximation as it does not include distortion due to the desired signal, such as intermodulation effects due to the desired signal.

CINR_A is used with the large signal VSA meter. The Source Signal input of the VSA meter is the signal with the desired input signal on (carrier+interference+noise). The Measured Signal input of the VSA meter is the signal with the desired signal off (interference+noise).

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
X-axis Test Point	System Meter	N/A
Channel fc	Real value	Frequency band of measured signal
Channel BW	Real value	Varies
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a computed C/(I+N) value for each sweep. By default the values are plotted as a function of swept channel power measured from the **X-axis Test Point** signal.

The C/(I+N) values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

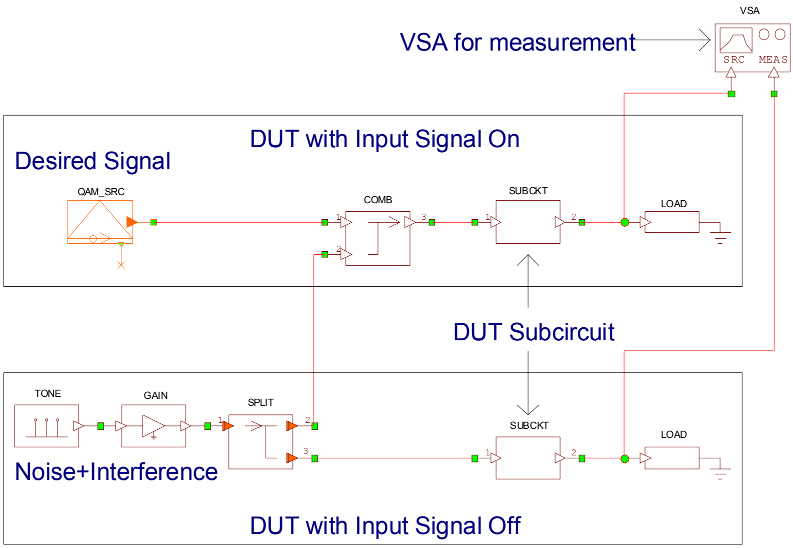
NOTE. The measurement will not return a value for a sweep if the noise+interference input is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The figure below illustrates one system diagram configuration for performing C/(I+N) measurements. The DUT is designed as a subcircuit to avoid having to duplicate the design. The splitter and combiner are used to introduce the same noise and interference signal to both DUTs.



The total power values for the two input signals are computed similar to the Power Meter measurement *PWR_MTR*.

If a measurement filter is selected through **Meas. Filter**, the filter is applied to the channel powers used to compute the C/(I+N) value, but not to the power assigned to the x-axis.

Swept Integrated Phase Noise: INTG_PHS_NOISE

Summary

INTG_PHS_NOISE computes integrated phase noise. The following output formats are available:

- dBc: The output is the power of the carrier in dBm minus the total power in a noise frequency band in dBm.
- Time Jitter, rms: The output is jitter in units of time, calculated from rms phase jitter according to:

$$TimeJitter = \frac{PhaseJitter}{2 \cdot \pi \cdot f_{Carrier}}$$

- Phase Jitter, rms: The output is jitter in units of angle, calculated from the integrated noise in dBc:

$$PhaseJitter = \sqrt{2 \cdot 10^{IntgNoise_{dBc}/10}}$$

The phase noise spectrum is generated from the time domain signal in a manner similar to the Phase Noise measurement *PHS_NOISE*.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA or Test Point	System Meter	N/A
Output Type	List of options	N/A
Start Freq. Offset	Real value	>0
End Freq. Offset	Real value	>0
Carrier Power	Real value	>0
Carrier Freq.	Real value	>0
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*RBW / # FFT Bins	Real value	>0

Name	Type	Range
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Start Offset	Real value	>=0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *Visual System Simulator Modeling Guide* for details.

Result

This measurement returns a computed integrated phase noise value for each sweep. When connected to a VSA or VNA_LS block, it displays the power at the Source Signal input of the block for the x-axis, using the **Input fc (x-axis)** and **Input BW** settings. When used with a TP block it displays a sequential number starting with 1 for the x-axis.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

NOTE. This measurement will not return a value for a sweep if the measured noise power is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The integrated phase noise is computed as the ratio of the power in a CW carrier to the power in a noise frequency band. The noise frequency band is defined by the **Start Freq. Offset** and **End Freq. Offset** settings, which specify frequency offsets from the carrier frequency. Negative offset values place the noise frequency band below the carrier, positive offset values place the noise frequency band above the carrier. The offsets must either both be positive or both be negative.

Care should be exercised when specifying both the frequency offset closest to the carrier and the RBW for the measurement. As explained in the *PHS_NOISE* measurement, applying a windowing function is necessary when measuring a signal with phase noise. However, the windowing functions typically have a relatively wide main lobe whose effect is to spread the power of the carrier over the main lobe. The width of the main lobe varies with the RBW/# FFT bins, with a larger number of FFT bins reducing the effective width of the main lobe.

The frequency offset near the carrier should be chosen so it falls outside this main lobe - otherwise a significant portion of the carrier power will be included in the measurement. If **RBW/#Bins** to “Auto”, the number of FFT bins will be chosen based on the specified frequency offset. However, for performance reasons the number of FFT bins is limited to 100,000 with this setting.

If both the frequency offset and **RBW/#Bins** setting are set to “Auto”, the frequency offset will be determined as in the PHS_NOISE measurement with the number of FFT bins set to 10,000.

Refer to *PHS_NOISE* for more information on the various measurement settings.

Swept Noise Factor/Figure (SNR In/SNR Out, Time Domain): NF_TD

Summary

NF_TD computes an estimate of noise factor/figure in a time domain simulation. Noise factor is computed by measuring the signal-to-noise ratios at the input and the output of the device under test (DUT). The noise factor is then:

$$F = \frac{SNR_{Input}}{SNR_{Output}}$$

Noise figure is simply $NF = 10 \cdot \log 10(F)$.

The signal-to-noise ratios are measured similar to the Swept Analog Signal to Noise Ratio measurement *SNR_A*.

NOTE. If only thermal noise is measured, take care in selecting the measured noise frequency band. If the measured noise frequency band contains frequency content such as interferers or distortion, that content is included in the estimated noise power.

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA).

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
X-axis Test Point	System Meter	N/A
Input Signal fc	Real value	Frequency band of measured signal
Output Signal fc	Real value	Frequency band of measured signal
Signal BW	Real value	Varies
Inp. Noise Offset	Real value	Varies
Out. Noise Offset	Real value	Varies

Name	Type	Range
Noise BW	Real value	Varies
*X-axis fc	Real value	Frequency band of reference signal
*X-axis BW	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*Signal VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a computed noise factor/figure value for each sweep. By default the values are plotted as a function of swept channel power measured from the **X-axis Test Point** signal.

You can display Noise figure by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

NOTE. The measurement will not return a value for a sweep if either the signal power at the output or the noise power at the input is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The signal-to-noise ratios at the input and output of the DUT are measured similar to the Swept Analog Signal to Noise Ratio measurement *SNR_A*.

The **Input Signal fc**, **Output Signal fc** and **Signal BW** settings define the signal frequency bands.

The **Noise Band Offset** and **Noise BW** settings define the measured noise frequency bands, which are relative to the center frequencies of the input and output signal frequency bands.

If a measurement filter is selected through **Meas. Filter**, the filter is applied to the measured powers used to compute the noise factor/figure value, but not to the power assigned to the x-axis.

Phase Noise: PHS_NOISE

Summary

PHS_NOISE computes the phase noise of a carrier signal in dBc/Hz versus frequency offset from the carrier.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Start Freq. Offset	Real value	>0
End Freq. Offset	Real value	>0
# of Freq.	Integer value	>0
Freq. Spacing	List of options	N/A
Carrier Power	Real value	>0
Carrier Freq.	Real value	>0
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Start Offset	Real value	>=0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value in units of dBc/Hz. The x-axis for this measurement is in frequency units.

The x-axis should normally be set to display a log scale when the **Freq. Spacing** setting is set to “Log Frequency”.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement computes phase noise by first computing the power spectral density (PSD) of the time domain signal using fast Fourier transforms (FFT). The PSD is computed similar to the Power Spectrum measurement *PWR_SPEC*.

The measurement determines the set of frequencies to be displayed from the **Start Freq. Offset**, **End Freq. Offset**, and **Number of Frequencies** settings. If the **Freq. Spacing** setting is set to “Log Frequency” the frequency offsets are computed using:

$$f_{off,\,i} = 10^{\log f_l + scale \cdot i} \qquad \text{for } i = 0, 1, \ldots, N-1$$

$$scale = \frac{(\log f_u - \log f_l)}{N-1}$$

where f_l is the smaller of **Start Freq. Offset** and **End Freq. Offset**, f_u is the larger of those offsets, and N is the **Number of Frequencies** setting.

The offset frequencies $f_{off,i}$ are then added to the carrier frequency to obtain absolute frequencies f_i. The output value for each absolute frequency fi is computed by first finding the FFT bin that contains the frequency.

If **Carrier Power** is set to “Not dBc” then the PSD value is output directly in dBm/Hz or dBW/Hz, otherwise it is converted to dBc/Hz. If the setting is “Auto” then the static signal power property from the signal is used as the carrier power. The PSD value is converted to dBc/Hz by subtracting the carrier power in dB from the PSD value in dB.

Note that because the FFT uses discrete frequency bins, it is possible, particularly in the lower frequency offsets, that multiple absolute frequencies share the same FFT bin. This results in a stepped output.

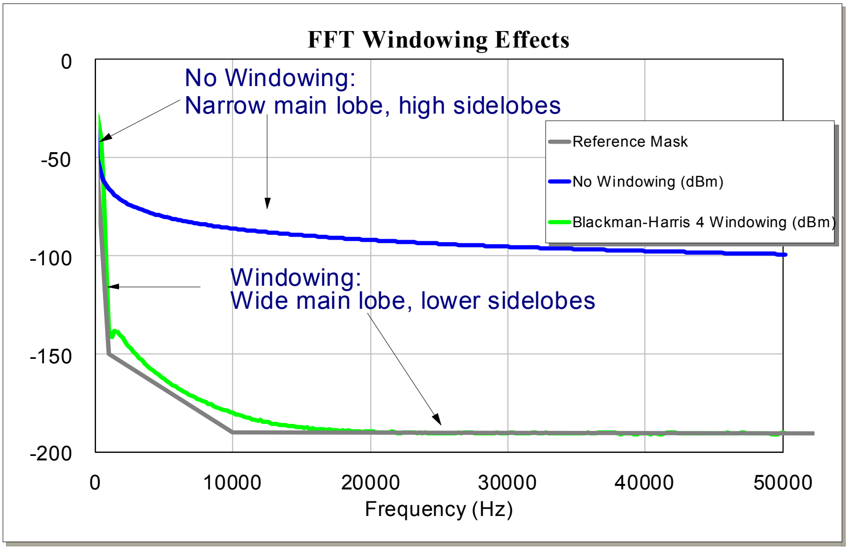
WINDOWING AND CUMULATIVE AVERAGING

By default the measurement is set to use a 10,000 point FFT, 4 term Blackman-Harris windowing and cumulative averaging. These settings were chosen to provide the best results for viewing the phase noise output for most cases.

Because a signal with phase noise is not periodic, performing an FFT directly on the signal results in a large amount of spectral leakage. The spectral leakage is the result of truncating the signal in order to perform the FFT. The power of the signal is leaked out over the entire frequency range, distorting the spectral estimate. This is most apparent at the higher frequency offsets from the carrier, where the true power level is fairly low and small amounts of leaked power have a proportionately larger effect on the estimate.

Leakage is commonly reduced by applying a window function to the time domain signal prior to performing the FFT. The window function typically tapers the start and end portions of the time domain signal. The effect on the FFT is to reduce the spectral leakage at the outer frequencies at the expense of a wider main lobe.

The following illustrates the effects of no windowing and windowing in an FFT generated spectrum when applied to a tone that has phase noise added. The gray bar is the phase mask used to generate the tone's phase noise.



Swept Analog Signal-to-Noise Ratio (Time Domain): SNR_A

Summary

SNR_A computes an estimate of the signal-to-noise ratio SNR in a time domain simulation. This measurement computes signal power in one frequency band and noise power in another frequency band. The average spectral density of the noise in the measured noise frequency band is then multiplied by the system bandwidth to obtain the total noise power in the system bandwidth. The output is then the signal power divided by the estimated total noise power in the system bandwidth.

NOTE. If only thermal noise is measured, take care in selecting the measured noise frequency band. If the measured noise frequency band contains frequency content such as interferers or distortion, that content is included in the estimated noise power.

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA).

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Signal fc	Real value	Frequency band of measured signal
Signal BW	Real value	Varies
System BW	Real value	Varies
Noise Band Offset	Real value	Varies
Noise BW	Real value	Varies
Frequency Handling	List of options	N/A
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*RBW / # FFT Bins	Real value	>0
*Signal VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1

Name	Type	Range
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the “Simulation Basics” chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a computed SNR value for each sweep. By default, values are plotted as a function of swept input channel power.

You can display the SNR values in dB by selecting the **dB** check box in the Add/ Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

NOTE. The measurement will not return a value for a sweep if the measured noise power is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The signal-to-noise ratio is computed as the ratio of the power in the signal measured in the signal frequency band to an estimate of the noise power in the system bandwidth.

The signal frequency band is defined by the **Signal fc** and **Signal BW** settings. The signal power is computed similar to the Power Meter (*PWR_MTR)* measurement.

The noise power in the system bandwidth is estimated from the average noise power in the measured noise frequency band. The noise frequency band is defined by the **Noise Band Offset** and **Noise BW** settings. The **Noise Band Offset** setting specifies the center of the noise frequency band relative to the center frequency of the signal frequency band. The noise power is computed similar to the Channel Power measurement with cumulative averaging, Hann windowing, and negative frequency folding in effect.

The **System BW** setting defines the system bandwidth. It can be set to a specified bandwidth, to match the signal bandwidth, or to match the noise bandwidth.

The noise power in the system bandwidth is estimated using:

$$N_{System} = \frac{N_{Measured}}{NoiseBW} \cdot SystemBW$$

If a measurement filter is selected through **Meas. Filter**, the filter is applied to the measured powers used to compute the SNR value, but not to the input power assigned to the x-axis.

Swept ACPR: ACPR

Summary

ACPR computes Adjacent Channel Power Ratio (ACPR) and plots it against a swept parameter (swept power by default).

This measurement should be used with either the Large Signal Vector Network Analyzer block VNA_LS or the Vector Signal Analyzer block VSA to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Channel fc	Real value	Frequency band of measured signal
Channel BW	Real value	Varies
Adj. Ch. Offset	Real value	Varies
Adj. Ch. BW	Real value	Varies
Frequency Handling	List of options	N/A
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a computed ACPR value for each sweep. When connected to a VSA or VNA_LS block, it displays the power at the Source Signal input of the block for the x-axis, using the **Input fc (x-axis)** and **Input BW** settings. When used with a TP block it displays a sequential number starting with 1 for the x-axis.

The ACPR values can be displayed in dB by selecting the **dB** check box in the Add/ Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

NOTE. The measurement will not return a value for a sweep if the measured channel power is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

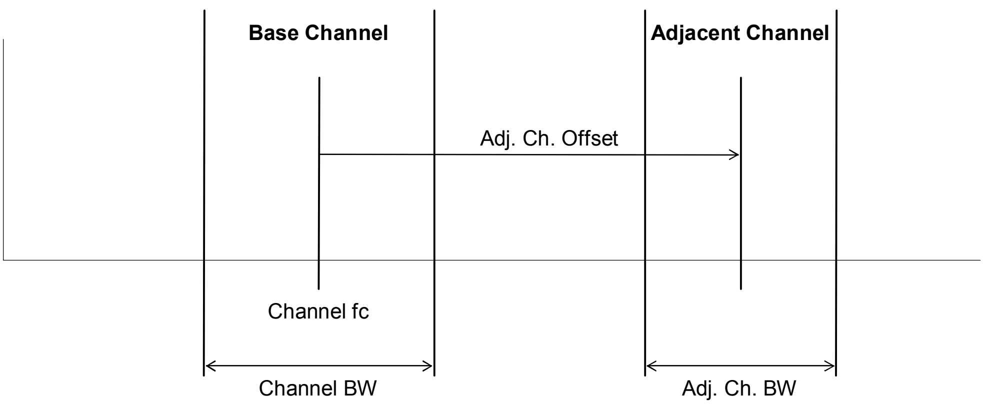
Computational Details

The adjacent channel power ratio is computed as the ratio of the power in the adjacent channel to the power in the base channel:

$$ACPR = \frac{P_{avg,\,adj}}{P_{avg,\,ch}}$$

where $P_{avg,adj}$ and $P_{avg,ch}$ are the average channel power in the adjacent and base channels, respectively. The channel powers are computed similar to the Power Meter measurement *PWR_MTR*.

The channel settings define the frequency ranges as illustrated in the following diagram:



If a measurement filter is selected through **Meas. Filter**, the filter is applied to the channel powers used to compute the ACPR value, but not to the input power assigned to the x-axis.

Instantaneous AM-AM: AMtoAM_INST

Summary

AMtoAM_INST computes instantaneous output power as a function of instantaneous input power.

This measurement should be used with the Vector Signal Analyzer block VSA or the Large Signal Vector Network Analyzer block VNA_LS in order to provide the reference and measured signals.

NOTE. AMtoAM_INST is generally not useful when the network possesses memory effects such as those due to filters or anti-aliasing resampling. The memory effects result in a skewing between the measured and reference signal values. In these cases the Swept Power AM-AM measurement ***AMtoAM_PS*** may provide a more useful measurement.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span Width	Real value	Varies
Time Span Units	List of options	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns the instantaneous output power level as a function of instantaneous input power level, sorted by the input power level. The values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. Both axes are in power units.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous power level is computed directly from the individual signal samples. The number of values displayed is determined by the **Time Span** and **Units** settings.

To avoid divide by zero errors, no output values are computed when the reference signal magnitude is below -200 dBV.

Swept AM to AM: AMtoAM_PS

Summary

AMtoAM_PS computes the AM-to-AM conversion and plots it against a swept parameter. By default the average output power is plotted against average input power.

This measurement should be used with either the Large Signal Vector Network Analyzer block VNA_LS or the Vector Signal Analyzer block VSA to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Output fc (y-axis)	Real value	Frequency band of measured signal
Output BW	Real value	Varies
Input fc (x-axis)	Real value	Frequency band of reference signal
Input BW	Real value	Varies
Frequency Handling	List of options	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns the average output channel power for each sweep. By default the values are plotted as a function of swept input channel power.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

The values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

This measurement computes the power in the reference and measured channels similar to the Power Meter measurement *PWR_MTR*.

If a measurement filter is selected through **Meas. Filter**, the filter is applied to the measured power value assigned to the y-axis, but not to the input power assigned to the x-axis.

Instantaneous AM-PM: AMtoPM_INST

Summary

AMtoPM_INST computes the instantaneous phase difference between a measured and a reference signal as a function of instantaneous input power.

This measurement should be used with the Vector Signal Analyzer block VSA or the Large Signal Vector Network Analyzer block VNA_LS in order to provide the reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span	Real value	Varies
Time Span Units	List of options	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns the instantaneous phase difference between the measured and reference signals as a function of instantaneous input power level, sorted by the input power level. The values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. Both axes are in power units.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous power level is computed directly from the individual signal samples. The number of values displayed is determined by the **Time Span** and **Units** settings.

To avoid arbitrary phase values due to round-off errors, no output values are computed when the reference signal magnitude is below -200 dB.

Swept AM to PM: AMtoPM_PS

Summary

AMtoPM_PS computes S21 phase and plots it against a swept parameter (swept power by default).

This measurement should be used with either the Large Signal Vector Network Analyzer block VNA_LS or the Vector Signal Analyzer block VSA to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Meas. Frequency	Real value	Frequency band of measured signal
Input fc (x-axis)	Real value	Frequency band of reference signal
Input BW	Real value	Varies
Frequency Handling	List of options	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns the computed S21 phase in angle units for each sweep. By default the values are plotted as a function of swept input channel power.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The phase is computed by subtracting the phase of the measured signal at the measurement frequency from the phase of the reference signal at the measurement frequency. To avoid arbitrary phase values due to round-off errors when the reference signal is very small, the reference phase is set to 0 if the magnitude of the reference signal is below -200 dBV. The phase values are computed similar to the Voltage Spectrum measurement *V_SPEC*. Input power is computed similar to the Power Meter measurement *PWR_MTR*.

Error Vector Magnitude: EVM

Summary

EVM computes the Error Vector Magnitude (EVM) between two signals.

This measurement should be used with either the Large Signal Vector Network Analyzer block VNA_LS or the Vector Signal Analyzer block VSA to provide the reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
EVM Metric	List of options	N/A
Interval	Integer Value	>0
Block Length	Real value	Varies
Final Output Type	List of options	N/A
Output Length	Integer Value	>0
Delay Compensation	List of options	N/A
Mag/Phase Compensation	List of options	N/A
Time Span	Real value	Varies
Start Offset	Real value	>= 0
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns unitless real values representing the computed EVM. The values are, with the exception of the "Absolute Error" EVM computation, expressed in percent. The EVM values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

Error Vector Magnitude computations are performed by first computing an error vector E that is the difference between the measured and reference signal. When attempting to measure modulation accuracy, the error vector is then typically sampled on the symbol boundaries for a block of symbols, and a metric is computed from those values. The metrics from several blocks may then be averaged. Alternatively, a simple time-domain waveform display of the error vector's magnitude may be desired.

There are numerous definitions for the EVM metric. The EVM measurement supports the following, selected through the **EVM Metric** setting:

"%RMS, SMax": The ratio of the root-mean square of the error to the maximum reference signal magnitude. The Digital Video Broadcasting specifications utilize this definition[1].

$$Metric = \frac{\sqrt{\frac{1}{N}\sum_{k \in K} E[k] \cdot E[k]^*}}{|S|_{Max,\,K}}$$

"%RMS, SAvg": The ratio of the root-mean square of the error to the root-mean square of the signal. The GSM/EDGE specifications utilize this definition[2].

$$Metric = \sqrt{\frac{\frac{1}{N}\sum_{k \in K} E[k] \cdot E[k]^*}{\frac{1}{N}\sum_{k \in K} S[k] \cdot S[k]^*}}$$

"%RMS, S[k]": The root-mean square of the ratios of the error to the signal.

$$Metric = \sqrt{\frac{1}{N}\sum_{k \in K}\frac{E[k] \cdot E[k]^*}{S[k] \cdot S[k]^*}}$$

"Absolute Error": The magnitude of the error vector. This is typically used to display the error magnitude as a time-domain waveform. To do so, set **Block Length** to 1 Value and **Output Length** to 1 block.

$$Metric = |E[k]|$$

$E[k]$ are the values from the error vector $E = Meas - S$ at the sample points, S is the reference signal, and K is the sampled set of points in the block being measured.

Note that the RMS based metrics require non-zero values for the reference signal. If $S[k]$ should happen to have a magnitude very close to 0 no EVM metric is generated for the block nor the final outputs that contain that block.

The **Interval** setting controls the spacing between the sampled values. The **Block Length** setting determines the overall length of each block. The first sample of the block corresponds to the first error/signal value, or k=0.

The metric values may then be output individually for each block, or they may be gathered over several blocks and further processed. The **Final Output Type** setting provides the following options:

"Average": The final output is the average of the metric values for each block.

"Peak": The final output is the metric value with the largest value in the block range. This is the peak of the averages.

The **Output Length** setting determines the number of blocks to incorporate in the final computation. The final output can be based on either a number of blocks or cumulative.

The EVM measurement also supports basic signal alignment compensation. The **Delay Compensation** setting determines the signal delay compensation performed. When set to "Basic", if either signal has had a signal delay introduced, an appropriate delay will be inserted into both signals to align the signals properly.

The **Mag/Phase Compensation** setting determines the adjustments to be made to the magnitude and phase of the measured signal. The "Basic" settings compute a static magnitude scaling and a phase rotation based on the static characteristics of the reference and measured signals. The measured signal is then adjusted by this scaling and/or rotation prior to computation of the error vector.

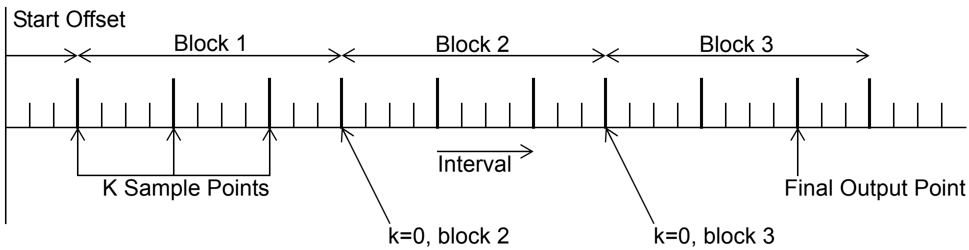
NOTE. The signal delay, static magnitude and static phase adjustments are determined prior to the actual signal generation, and may be only rough approximations. They are determined by blocks processing the signals. These adjustments are comparable to the basic delay, gain and phase compensation used by most of the receiver blocks in VSS.

Also note that the static magnitude scaling requires an RF based signal such as one generated by the TONE or NTONE source blocks, or by a transmitter.

The **Time Span** setting controls the number of output values displayed.

The **Start Offset** setting determines the location of the sample points relative to the start of the measured signal. If it is set to "Auto", the start offset will be 0 if the signal is not oversampled. If the signal is oversampled, the start offset will be set to the mid-point of the symbol, or 0.5 Symbols.

The following illustrates some of the concepts for an interval setting of 4 samples, 3 blocks, and a start offset of 3 samples:



Notes:

- If the block length is not a multiple of the interval spacing, the start of the following block will not fall on the same symbol boundaries as the previous block.
- The time associated with the output value (the x-axis value) is the time of the last sampled point in the last block.

References

[1] 3GPP TS 45.005 V5.2.0 (2001-11), Annex G.

[2] ETSI TR101 290 V1.2.1 (2001-05), Digital Video Broadcasting (DVB); Measurement guidelines for DVB systems.

Swept Error Vector Magnitude: EVM_PS

Summary

EVM_PS computes the Error Vector Magnitude (EVM) between two signals and plots it against a swept parameter (swept power by default).

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA) to provide the reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
X-axis Test Point	System Meter	N/A
EVM Metric	List of options	N/A
Interval	Integer Value	>0
Block Length	Real value	Varies
Final Output Type	List of options	N/A
Output Length	Integer Value	>0
Delay Compensation	List of options	N/A
Mag/Phase Compensation	List of options	N/A
Start Offset	Real value	>= 0
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*Input Frequency Handling	List of options	N/A
*Input RBW / # FFT Bins	Real value	>0
*Input VBW / # Avg.	Real value	>0

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns the computed EVM per sweep. By default the EVM is plotted as a function of swept channel power measured from the **X-axis Test Point** signal.

The EVM values are, with the exception of the "Absolute Error" EVM computation, expressed in percent. The EVM values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

NOTE. The measurement will not return a value for a sweep if the denominator for the selected metric computation is very close to zero.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

Error Vector Magnitude computations are performed by first computing an error vector E that is the difference between the measured and reference signal. When attempting to measure modulation accuracy, the error vector is then typically sampled on the symbol boundaries for a block of symbols, and a metric is computed from those values. The metrics from several blocks may then be averaged.

There are numerous definitions for the EVM metric. The EVM measurement supports the following, selected in **EVM Metric**:

- "%RMS, SMax": The ratio of the root-mean square of the error to the maximum reference signal magnitude. The Digital Video Broadcasting specifications utilize this definition[1].

$$Metric = \frac{\sqrt{\frac{1}{N}\sum_{k \in K} E[k] \cdot E[k]^*}}{|S|_{Max,\,K}}$$

- "%RMS, SAvg": The ratio of the root-mean square of the error to the root-mean square of the signal. The GSM/EDGE[1], 802.11a[3] and 802.16[4] and specifications utilize this definition.

$$Metric = \sqrt{\frac{\frac{1}{N}\sum_{k \in K} E[k] \cdot E[k]^*}{\frac{1}{N}\sum_{k \in K} S[k] \cdot S[k]^*}}$$

- "%RMS, S[k]": The root-mean square of the ratios of the error to the signal.

$$Metric = \sqrt{\frac{1}{N}\sum_{k \in K} \frac{E[k] \cdot E[k]^*}{S[k] \cdot S[k]^*}}$$

- "Absolute Error": The magnitude of the error vector. This is typically used to display the error magnitude as a time-domain waveform. To do so, set **Block Length** to 1 Value and **Output Length** to 1 block.

$$Metric = |E[k]|$$

E[*k*] are the values from the error vector *E* = *Meas* - *S* at the sample points, *S* is the reference signal, and *K* is the sampled set of points in the block being measured.

Note that the RMS-based metrics require non-zero values for the reference signal. If *S*[*k*] should happen to have a magnitude very close to 0, no EVM metric is generated for the block nor the final outputs that contain that block.

Interval controls the spacing between the sampled values. **Block Length** determines the overall length of each block. The first sample of the block corresponds to the first error/signal value, or k=0.

The metric values may then be output individually for each block, or they may be gathered over several blocks and further processed. **Final Output Type** provides the following options:

- "Average": The final output is the average of the metric values for each block.
- "Peak": The final output is the metric value with the largest value in the block range. This is the peak of the averages.

Output Length determines the number of blocks to incorporate in the final computation. The final output can be based on either a number of blocks or cumulative.

The EVM measurement also supports basic signal alignment compensation. **Delay Compensation** determines the signal delay compensation performed. When set to "Basic", if either signal has had a signal delay introduced, an appropriate delay is inserted into both signals to align the signals properly.

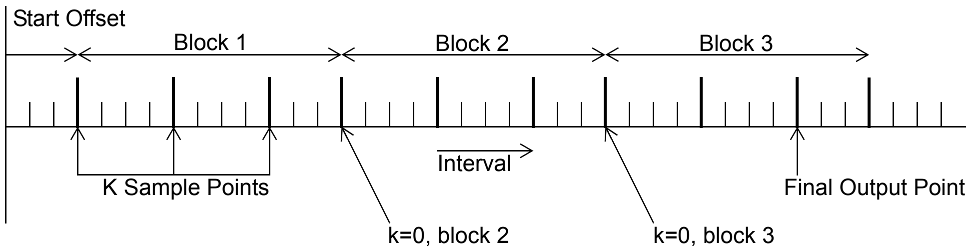
Mag/Phase Compensation determines the adjustments to be made to the magnitude and phase of the measured signal. The "Basic" settings compute a static magnitude scaling and a phase rotation based on the static characteristics of the reference and measured signals. The measured signal is then adjusted by this scaling and/or rotation prior to computation of the error vector.

NOTE. The signal delay, static magnitude and static phase adjustments are determined prior to the actual signal generation, and may be only rough approximations. They are determined by blocks processing the signals. These adjustments are comparable to the basic delay, gain and phase compensation used by most of the receiver blocks in VSS.

Also note that the static magnitude scaling requires an RF based signal such as one generated by the TONE source block or by a transmitter.

The **Start Offset** setting determines the location of the sample points relative to the start of the measured signal. If it is set to "Auto", the start offset will be 0 if the signal is not oversampled. If the signal is oversampled, the start offset will be set to the mid-point of the symbol, or 0.5 Symbols.

The following illustrates some of the concepts for an interval setting of 4 samples, 3 blocks, and a start offset of 3 samples:



Notes:

- If the block length is not a multiple of the interval spacing, the start of the following block will not fall on the same symbol boundaries as the previous block.

References

[1] ETSI TR101 290 V1.2.1 (2001-05), Digital Video Broadcasting (DVB); Measurement guidelines for DVB systems.

[2] 3GPP TS 45.005 V5.2.0 (2001-11), Annex G.

[3] IEEE Std 802.11a-1999, "Part 11: Wireless LAN Medium Access Control (MAC) and Physical Layer (PHY) specifications, High-speed Physical Layer in the 5 GHz Band"

[4] IEEE Std 802.16-2004, "Part 16: Air Interface for Fixed Broadband Wireless Access Systems"

Instantaneous Error Vector Spectrum: EVS

Summary

EVS computes the instantaneous Error Vector Spectrum (EVS) between two signals. The measurement works with the Vector Signal Analyzer meter block VSA or the Large Signal Vector Network Analyzer block VNA_LS. The EVS is the spectrum of the vector differences between the measured and source inputs to the VSA meter.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span	Real value	Varies
Time Span Units	List of options	N/A
Number Averages	Integer value	>0

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns complex values in units of voltage. The voltage can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units. The range of the frequency axis is $f_c - f_s/2 \le f < f_c + f_s/2$, where f_c is the center frequency of the measured signal and f_s is the sampling frequency.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous error vector spectrum is computed based on the following:

$$V[f_k] = \frac{\sum_{i=0}^{N-1} w_i E_i \exp(-j2\pi ik/N)}{\sum_{i=0}^{N-1} w_i}, \qquad k = 0, 1, \ldots, N-1$$

where $V[f_k]$ is the amplitude of the f_k frequency, f_s is the sampling frequency of the measured signal, and N is the number of FFT bins. E_i is the sequence of error vectors and w_i is an optional windowing function. For $w_i = 1, i = 0, 1, \ldots, N-1$ (no windowing), the spectrum equation is the discrete Fourier transform.

The error vectors are computed as:

$$E[k] = Meas[k] - Src[k]$$

where $Meas$ and Src are the two complex signal inputs to the VSA meter block.

The number of FFT bins is determined by the **Time Span** and **Units** settings, and is set to the equivalent number of samples in the data window.

The spectrum may optionally be computed from an average of several spectrum computations. This occurs when the number of averages is set to a value other than 1. When averaging is in effect, individual spectrums are computed by computing spectrums for several windows of input data, each offset by 50% from the previous data window. A Taylor window function is applied to the data before each FFT is performed. The average of all the spectrums is then used to compute the spectrum values.

Small Signal Group Delay: GD_SS

Summary

GD_SS computes the group delay of linear networks. The measurement works with the Small Signal Vector Network Analyzer block VNA_SS. This measurement can be used in the same graph with the Microwave Office "GD" measurement.

NOTE. GD_SS and VNA_SS should only be used with linear RF blocks. Nonlinear and non-RF blocks will produce incorrect results.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Small Signal VNA (VNA_SS)	System VNA_SS Meter	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns real values in units of time.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The group delay time is estimated from the phase components of the frequency response using the following:

$$GD[f_k] = -\frac{(\theta[f_{k+1}] - \theta[f_k]) + (\theta[f_k] - \theta[f_{k-1}])}{2 \cdot 2\pi\Delta f} \qquad \text{if } 1 < k < N$$

$$GD[f_1] = -(\theta[f_2] - \theta[f_1])/(2\pi\Delta f) \qquad \text{if } k = 1$$

$$GD[f_N] = -(\theta[f_N] - \theta[f_{N-1}])/(2\pi\Delta f) \qquad \text{if } k = N$$

where N is the number of frequency bins used to generate the input signal's FFT, $\theta[f_k]$ is the phase at the k'th frequency, and Δf is the frequency spacing between FFT bins. The phase differences are adjusted to fall within +/- π. The number of FFT bins is determined by the VNA_SS block.

The measurement computes the FFT using N samples starting at the first sample after any static signal delay in the signal.

Swept Intermodulation Distortion (dBc): IMD

Summary

IMD computes an nth-order intermodulation distortion measurement of a VSS RF circuit. The measurement outputs the power of the intermodulation product relative to the power of a single fundamental component.

For a two-tone analysis, the order n of the intermodulation product is given by:

$$n = |h_1| + |h_2|$$

where h_1 is the harmonic of the first tone and h_2 is the harmonic of the second tone. The intermodulation product $2f_1 - f_2$ is therefore a 3rd order product.

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA) to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
IM Order	Integer value	>1
Fund. Output Freq.	Real value	Frequency band of measured signal
IM Frequency	Real value	Frequency band of measured signal
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A

Name	Type	Range
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *Visual System Simulation Modeling Guide* for details.

Result

This measurement returns the **IM Order**'th intermodulation distortion in dBc relative to the power in **Fund. Output Freq**. When connected to a VSA or VNA_LS block, it displays the power at the Source Signal input of the block for the x-axis, using the **Input fc (x-axis)** and **Input BW** settings. When used with a TP block it displays a sequential number starting with 1 for the x-axis.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The default settings of the measurement configure the measurement for use with a one or two-tone signal. When the signal is a one or two-tone signal, the measurement can automatically determine the fundamental and IM frequencies. The frequencies can also be explicitly specified individually. If the signal is neither a one-tone nor a two-tone signal the frequencies must be explicitly specified.

For one-tone signals, the measurement uses the tone and the appropriate harmonic. Note that for one-tone signals the sampling frequency band must be wide enough to include the appropriate harmonic.

For two-tone signals, the measurement determines the frequencies according to the following:

Fundamental Output Frequency: The frequency of the tone with the smaller output power is selected.

IM Frequency: The frequency is determined from the following:

$$f_{IM,\,Lower} = |h_1 \cdot f_1 + h_2 \cdot f_2|$$

$$f_{IM,\,Upper} = |h_2 \cdot f_1 + h_1 \cdot f_2|$$

where f_1 and f_2 are the specified tones, and the corresponding harmonics computed from:

$$h_1 = -n/2 \qquad \text{for } n \text{ even}$$

$$h_1 = -(n-1)/2 \qquad \text{for } n \text{ odd}$$

$$h_2 = n + h_1$$

For odd n, the above harmonics result in IM products adjacent to the main signal. The frequency of the IM product with the larger power is selected.

The selection of the output fundamental tone with the lower power and the IM product with the higher power level results in the IMD measurement with the smallest, and therefore more conservative, value.

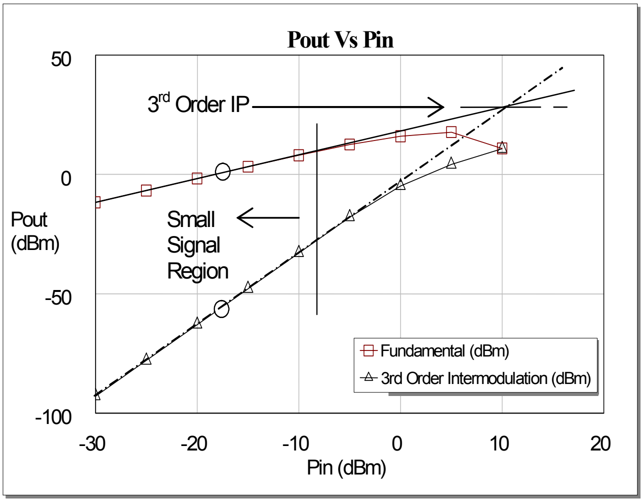
FREQUENCY RESTRICTIONS

The fundamental and IM frequencies must all fall within the sampling frequency band of the signal. If any of the frequencies fall outside of the band, an error is reported. Different frequencies should either be selected or the sampling frequency increased.

Swept nth-order Intercept Point: IPn

Summary

IPn computes the nth-order intercept point of a VSS RF circuit. The intercept point is the point at which a linear extrapolation of the fundamental power and the power in the intermodulation product intersect each other when shown as output power in dBm versus input power in dBm. The following figure shows an example of the third-order intercept point of a two-tone excitation.



The nth-order intercept point should be performed in the small-signal operating region of the device. In the previous example, the pair of points marked with circles is a good choice for determining the intercept point. In the small-signal region, the slope of the IM curve is n, the order of the product. Because the slopes of both curves are known, a measurement at a single power value is sufficient to determine the intercept point (assuming the point is in the small signal region).

The output intercept point is given by:

$$OIP = PF_o + \left(\frac{PF_0 - PN_o}{n-1}\right)$$

where PF_o is the output power of the fundamental component in dBm, PN_o is the output power of the nth order product, and n is the order.

The input intercept point is similarly given by:

$$IIP = PF_i + \left(\frac{PF_0 - PN_o}{n-1}\right)$$

where PF_i is the input power of the fundamental component in dBm.

For example, for a two-tone analysis, the order, n, of the intermodulation product is given by:

$$n = |h_1| + |h_2|$$

where h_1 is the harmonic of the first tone and h_2 is the harmonic of the second tone. The intermodulation product $2f_1 - f_2$ is a 3rd order product.

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA) to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Intercept Order	Integer value	>1
IP Type	List of options	N/A
Fund. Output Freq.	Real value	Frequency band of measured signal
IM Frequency	Real value	Frequency band of measured signal

Name	Type	Range
Fund. Input Freq.	Real value	Frequency band of reference signal
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a computed **Intercept Order**'th intercept point value for each sweep. **IP Type** determines whether the intercept point is input power or output power. By default the values are plotted as a function of swept input channel power.

The intercept point values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The default settings of the measurement configure the measurement for use with a one or two-tone signal. When the signal is a one or two-tone signal, the measurement can automatically determine the fundamental and IM frequencies. The frequencies can also be explicitly specified individually. If the signal is neither a one-tone nor a two-tone signal the frequencies must be explicitly specified.

For one-tone signals, the measurement uses the tone and the appropriate harmonic, adjusting the IPn computation because a harmonic is used rather than an IM product. Note that for one-tone signals the sampling frequency band must be wide enough to include the appropriate harmonic.

For two-tone signals, the measurement determines the frequencies according to the following:

Fundamental Output Frequency: The frequency of the tone with the smaller output power is selected.

IM Frequency: The frequency is determined from the following:

$$f_{IM,\,Lower} = |h_1 \cdot f_1 + h_2 \cdot f_2|$$

$$f_{IM,\,Upper} = |h_2 \cdot f_1 + h_1 \cdot f_2|$$

where f_1 and f_2 are the specified tones, and the corresponding harmonics computed from:

$$h_1 = -n/2 \qquad \text{for } n \text{ even}$$

$$h_1 = -(n-1)/2 \qquad \text{for } n \text{ odd}$$

$$h_2 = n + h_1$$

For odd n, the above harmonics result in IM products adjacent to the main signal. The frequency of the IM product with the larger power is selected.

Fundamental Input Frequency: If an output fundamental frequency is explicitly specified, that frequency is used. Otherwise, the input frequency of the tone corresponding to the selected output fundamental is used.

The selection of the output fundamental tone with the lower power and the IM product with the higher power level results in the OIPn measurement with the smallest, and therefore more conservative, value.

FREQUENCY RESTRICTIONS

The fundamental and IM frequencies must all fall within the sampling frequency band of the signal. If any of the frequencies fall outside of the band, an error is reported. Different frequencies should either be selected or the sampling frequency increased.

Instantaneous IQ Error Magnitude: IQERRMAG

Summary

IQERRMAG computes the instantaneous error between the magnitudes of two signal. The measurement works with the Vector Signal Analyzer block VSA or the Large Signal Vector Network Analyzer block VNA_LS. The error is computed from the difference between the magnitudes of the measured and source inputs to the VSA meter.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span	Real value	Varies
Time Span Units	List of options	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

* *indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns unitless real values. The error values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous IQ error magnitude is computed according to the following:

$$IQErrMag\{t\} = abs(|Meas[t]| - |Src[t]|)$$

where $Meas$ and Src are the two complex signal inputs to the VNA_LS/VSA block.

Instantaneous IQ Error Phase: IQERRPHS

Summary

IQERRPHS computes the instantaneous error between the phases of two signal. The measurement works with the Vector Signal Analyzer block VSA or the Large Signal Vector Network Analyzer block VNA_LS. The error is computed from the difference between the phases of the measured and source inputs to the VSA meter.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span	Real value	Varies
Time Span Units	List of options	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns real values in angular units. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** Units settings.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous IQ error phase is computed according to the following:

$$IQErrMag[t] = abs(\angle Meas[t] - \angle Src[t])$$

where $Meas$ and Src are the two complex signal inputs to the VNA_LS/VSA block. The result is adjusted to satisfy $IQErrMag[t] \leq \pi$.

Swept S21: S21_PS

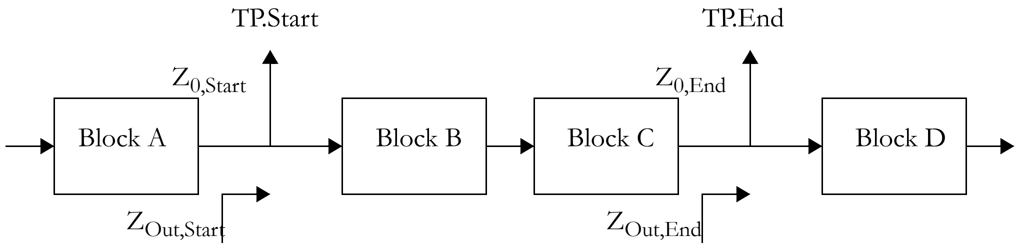
Summary

S21_PS computes S21 in a VSS time domain simulation and plots it against a swept parameter (swept input power by default).

S21 is computed as the ratio of the reflected voltage at the output (End) to the incident voltage at the input (Start) of a DUT:

$$S21 = \frac{V^{-}_{End}}{V^{+}_{Start}} = \frac{V_{End}}{V_{Start}} \cdot \frac{\dfrac{Z_{Out,\,End} + Z_{0,\,End}}{2 \cdot Z_{Out,\,End}}}{\dfrac{Z_{Out,\,Start} + Z_{0,\,Start}}{2 \cdot Z_{Out,\,Start}}} \cdot \sqrt{\frac{Re\{Z_{0,\,Start}\}}{Re\{Z_{Out,\,End}\}}}$$

where the various impedances are:



In the above diagram, $Z_{0,Start}$ is the characteristic impedance of the output of Block A and $Z_{Out,Start}$ is the impedance seen looking out the output of Block A. $Z_{0,End}$ is the characteristic impedance of the output of Block C and $Z_{Out,End}$ is the impedance seen looking out the output of Block C.

The voltage gain V_{End}/V_{Start} is similar to the voltage gain measured by the Swept Voltage Gain V_GAIN.

NOTE. If impedance mismatch modeling is not enabled, the results will be identical to the Swept Voltage Gain measurement, as the impedances will all be identical.

This measurement should be used with either the Large Signal Vector Network Analyzer block VNA_LS or the Vector Signal Analyzer block VSA to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Meas. Frequency	Real value	Frequency band of measured signal
Input Frequency	Real value	Frequency band of reference signal
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a complex value. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. The value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in power units.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The S21 value is computed by dividing the measured signal's value at the measurement frequency by the reference signal's value at the measurement frequency and applying corrections based on impedance mismatches. To avoid divide by zero and arbitrary phase values due to round-off errors, S21 values are not computed if the magnitude of the reference signal is below -200 dBV.

The measurement frequencies are determined by the **Meas. Frequency** and **Input Freq.** settings. If **Meas. Frequency** is set to "Auto" then the measurement frequency at the output is determined from the signal. If the signal is CW then it is set to the frequency of the first fundamental, otherwise it is set to the center frequency of the signal.

Input Freq. is normally set to "Output Freq." when the measured frequency is the same for both the input and output voltages. When "Output Freq." is set, the input frequency will be chosen in a manner similar to the output frequency. If the output frequency is explicitly set, then the input frequency will be set to the same frequency. If **Meas Frequency** is set to "Auto", then the input frequency will be set using the same algorithm but applied to the reference input signal.

The **Input FC (x-axis)** and **Input BW (x-axis)** settings are only used for generating input power values for the x-axis when an explicit swept variable has not been selected for the x-axis.

The values at the measurement frequency are computed similar to the Voltage Spectrum measurement *V_SPEC*. Input power is computed similar to the Power Meter measurement *PWR_MTR*.

Small Signal Frequency Response: S21_SS

Summary

S21_SS computes the frequency response of linear networks. The measurement works with the Small Signal Vector Network Analyzer block VNA_SS.

NOTE. S21_SS and VNA_SS should only be used with linear RF blocks. Nonlinear and non-RF blocks will produce incorrect results.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Small Signal VNA (VNA_SS)	System VNA_SS Meter	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns complex values. The absolute value can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Small Signal Impulse Response: TD_SS

Summary

TD_SS computes the impulse response (only S21) of linear networks. The measurement works with the Small Signal Vector Network Analyzer block VNA_SS.

NOTE. GD_SS and VNA_SS should only be used with linear RF blocks. Nonlinear and non-RF blocks will produce incorrect results.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Small Signal VNA (VNA_SS)	System VNA_SS Meter	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns complex values in units of voltage. The voltage can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in samples.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Swept Voltage Gain: V_GAIN

Summary

V_GAIN computes voltage gain in a VSS time domain simulation and plots it against a swept parameter (swept input power by default).

The voltage gain is computed as:

$$G_V = \frac{V_{Output}}{V_{Input}}$$

where V_{Output} and V_{Input} are the measured voltages at the output and input of the DUT.

This measurement should be used with either the Large Signal Vector Network Analyzer block (VNA_LS) or the Vector Signal Analyzer block (VSA) to provide the swept reference and measured signals.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Output Freq.	Real value	Frequency band of measured signal
Input Freq.	Real value	Frequency band of reference signal
Gain Type	List of options	N/A
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW (x-axis)	Real value	Varies
*Frequency Handling	List of options	N/A
*RBW / # FFT Bins	Real value	>0
*VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A

Name	Type	Range
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a complex value. This complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. The value can also be displayed in dB by selecting the **dB** checkbox in the Add/Modify Measurement dialog box. The x-axis for this measurement is in power units.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The voltage gain is computed by dividing the measured signal's value at the measurement frequency by the reference signal's value at the measurement frequency. To avoid divide by zero and arbitrary phase values due to round-off errors, gain values are not computed if the magnitude of the reference signal is below -200 dBV.

The measurement frequencies are determined by the Output Freq. and Input Freq. settings. If Output Freq. is set to "Auto" the measurement frequency at the output is determined from the signal. If the signal is CW then it is set to the frequency of the first fundamental, otherwise it is set to the center frequency of the signal.

Input Freq. is normally set to "Output Freq." when the measured frequency is the same for both the input and output voltages. When “Output Freq.” is set, the input frequency is chosen in a manner similar to the output frequency. If the output frequency is explicitly set, then the input frequency is set to the same frequency. If Output Freq. is set to “Auto”, then the input frequency is set using the same algorithm but applied to the reference input signal.

The Input FC (x-axis) and Input BW (x-axis) settings are only used for generating input power values for the x-axis when an explicit swept variable has not been selected for the x-axis.

The values at the measurement frequency are computed similar to the Voltage Spectrum measurement *V_SPEC*. Input power is computed similar to the Power Meter measurement *PWR_MTR*.

Instantaneous S21: V_GAIN_INST

Summary

V_GAIN_INST computes instantaneous S21 as a function of instantaneous input power.

This measurement should be used with the Vector Signal Analyzer meter block VSA or the Large Signal Vector Network Analyzer meter block VNA_LS in order to provide the reference and measured signals.

NOTE. V_GAIN_INST is generally not useful when the network possesses memory effects such as those due to filters or anti-aliasing resampling. The memory effects result in a skewing between the measured and reference signal values. In these cases the Swept Voltage Gain measurement *V_GAIN* may provide a more useful measurement.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA (VNA_LS/VSA)	System VNA_LS meter, System VSA meter	N/A
Time Span	Real value	Varies
Time Span Units	List of options	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a complex value. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the measurement dialog. The value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

The x-axis for this measurement is in power units. The values are sorted by the input power level.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The instantaneous power level is computed directly from the individual signal samples. The number of values displayed is determined by the **Time Span** and **Units** settings.

The S21 value is computed as the ratio of the measured signal samples to the reference signal samples. To avoid arbitrary phase values due to round-off errors and divide by zero errors, no output values are computed when the reference signal magnitude is below -200 dBV.

Instantaneous Power vs. Time: PWR_INST

Summary

PWR_INST plots instantaneous power versus time.

The measurements in the Power category differ in the following ways:

- *PWR_vsT* - Computes average power versus time with an optional frequency band.
- *PWR_MTR* - Computes average power within an optional frequency band. The x-axis may be another input power or a swept variable.
- PWR_INST - Computes instantaneous power versus time.

PWR_MTR provides the most flexibility when working with output equations.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Time Span	Real value	>0
Time Span Units	List of options	N/A
Wait for Full Window	Check box	N/A
Start Offset	Real value	>= 0
Signal Delay	List of options	N/A
X Axis Units	List of options	N/A
X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

The measurement returns real values. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** and **Units** settings.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

If the signal is modulated (center frequency is not 0) it plots:

$$y = \frac{|v|^2}{2Z_0}$$

otherwise it plots:

$$y = \frac{|v|^2}{Z_0}$$

where v is the measured signal and Z_0 is the default impedance from the System Simulator Options dialog box.

Power Meter (Supports Sweeping): PWR_MTR

Summary

PWR_MTR computes average power in a time domain simulation. A frequency band may be specified. Power may be displayed as total power or power spectral density (power/Hz).

PWR_MTR can display power versus the swept variable when used with swept simulations. PWR_MTR can also display power versus measured input power when used with the Vector Signal Analyzer block VSA or the Large Signal Vector Network Analyzer block VNA_LS.

The measurements in the Power category differ in the following ways:

- PWR_vsT - Computes average power versus time with an optional frequency band.
- PWR_MTR - Computes average power within an optional frequency band. The x-axis may be another input power or a swept variable.
- PWR_INST - Computes instantaneous power versus time.

PWR_MTR provides the most flexibility when working with output equations.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Large Signal VNA/VSA or Test Point	System Meter	N/A
Band Center	Real value	Frequency band of measured signal
Bandwidth	Real value	Varies
Output Type	List of options	N/A
*Input fc (x-axis)	Real value	Frequency band of reference signal
*Input BW	Real value	Varies
Frequency Handling	List of options	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
*Overlap Ratio for Averaging	Real value	<=1

Name	Type	Range
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns the average output channel power for each sweep. When connected to a VSA or VNA_LS block, it displays the power at the Source Signal input of the block for the x-axis, using the **Input fc (x-axis)** and **Input BW** settings. When used with a TP block it displays a sequential number starting with 1 for the x-axis.

The values can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The measured power is computed using the discrete power spectrum of the measured waveform. If average power is being computed, a series of power spectrums is used. The average may be computed as a running average or a cumulative average.

The discrete power spectrum is computed similar to the Power Spectrum measurement *PWR_SPEC* with **Spectrum Type** set to **Power Spectrum (Pwr/Bin)**.

The **Frequency Handling** setting is similar to PWR_SPEC's **Frequencies Displayed** setting, and determines how negative frequencies are handled. If negative frequencies are folded, the channel frequency band includes both positive and negative frequency components. If negative frequencies are not folded, only the frequency components falling within the actual frequency band are included.

Channel power is then computed from the discrete power spectrum by summing the power values corresponding to the frequency bins contained within the channel frequency range. If a channel edge frequency is only partially contained within a bin, the power value corresponding to the bin is included if the frequency range extends over at least half the bin.

For example, suppose the measured signal has a center frequency of 10GHz and a sampling frequency of 10GHz. It contains three tones:

5 mW at 7.9 GHz

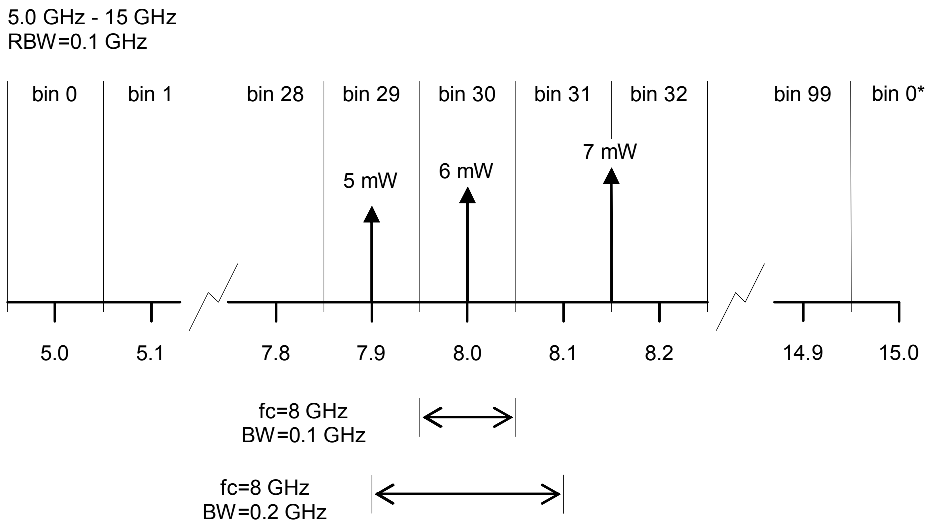
6 mW at 8.0 GHz

7 mW at 8.15 GHz

Assume that RBW has been set to 0.1 GHz. This results in an FFT containing 100 bins (10 GHz/0.1 GHz). Because the bins are always centered on the center frequency, the bin boundaries fall on the 0.05 GHz fraction, or ..., 7.85, 7.95, 8.05, ...

If the channel center frequency is set to 8 GHz and the channel bandwidth is set to 0.1 GHz, the channel band is 7.95 to 8.05 GHz, which is the single bin containing the 8 GHz tone. The measurement returns a channel power of 6 mW.

If the channel center frequency is set to 8 GHz and the channel bandwidth is set to 0.2 GHz, the channel band is 7.9 to 8.1 GHz, which is half of the bin containing the 7.9 GHz tone, the full bin containing the 8 GHz and half of the bin centered at 8.1 GHz. Because the frequency range covers at least half of the bin containing the 7.9 GHz tone, its power contribution is included. The bin centered at 8.1 GHz is also covered by at least half the frequency range, so its power would be included. Due to the nature of the FFT, the tone at 8.15 GHz, which falls on a bin boundary, is entirely within the following bin, bin 32, and not in the preceding bin, bin 31. The measurement then returns a channel power of 11 mW.



Note that because the power measurements are performed using FFTs of the time-domain waveform, the simulation times when the measurements are generated are determined by several factors. The following determine how many data samples are required before a measurement can be computed.

If no averaging:

$$\#Samples_i = N \cdot i$$

If averaging:

$$\#Samples_i = N_{avg}N - (N_{avg} - 1)(1 - S_{ratio}) + S_{ratio}(i - 1)$$

where $\#Samples_i$ is the number of samples required for the i'th measurement, N_{avg} is the number of averages, and S_{ratio} is the sliding ratio between windows, where 1 represents a full data window width (0% overlap). The i'th measurement is output at simulation time $t_i = \#Samples_i / f_s$.

You can use **Meas. Filter** and **Alpha** to apply a measurement filter to the power spectrum prior to the computation of the channel power. The filter is applied in the frequency domain to the power spectrum after any averaging. The following filters are supported:

- Root-raised cosine:

$$
H[f] = \begin{cases} 1 & \left(0 \le |f - f_{ch}| \le \dfrac{1-\alpha}{2T}\right) \\ \dfrac{1}{2}\left(1 + \cos\left(\dfrac{\pi T}{\alpha}\left(|f - f_{ch}| - \dfrac{1-\alpha}{2T}\right)\right)\right) & \left(\dfrac{1-\alpha}{2T} < |f - f_{ch}| \le \dfrac{1+\alpha}{2T}\right) \\ 0 & \left(\dfrac{1+\alpha}{2T} < |f - f_{ch}|\right) \end{cases}
$$

where $T = 1/\text{Channel BW}$ and $f_{ch} = \text{Channel fc}$.

If a measurement filter is selected through **Meas. Filter**, the filter is applied to the measured power value assigned to the y-axis, but not to the input power assigned to the x-axis, if any.

The **Start Offset** setting lets you skip over an initial segment of the signal. This is particularly useful when using cumulative averaging with a signal that has a considerable initial transient.

The **Signal Delay** setting lets you control whether delays imparted upon the signal by blocks such as DELAY, DLY_SMP or DLY_SYM, or filters, are included in the measurement. The delay is determined by the propagated signal delay property.

Channel Power vs. Time: PWR_vsT

Summary

PWR_vsT displays channel power versus time.

The measurements in the Power category differ in the following ways:

- PWR_vsT - Computes average power versus time with an optional frequency band.
- *PWR_MTR* - Computes average power within an optional frequency band. The x-axis may be another input power or a swept variable.
- *PWR_INST* - Computes instantaneous power versus time.

PWR_MTR provides the most flexibility when working with output equations.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Channel Center	Real value	Varies
Channel BW	Real value	> 0
Frequency Handling	List of options	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
Time Span	Real value	> 0
Time Span Units	List of options	N/A
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Meas. Filter	List of options	N/A
*Alpha	Real value	Depends on Meas. Filter
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A
*X Axis Units	List of options	N/A
*X Axis Start	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the “Simulation Basics” chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns real values in power units. The power can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis is scrollable simulation time. The width of the x-axis is determined from the **Time Span** value, which, depending on the units setting, uses the samples per symbol from the measured waveform.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

Channel power is computed using the discrete power spectrum of the measured waveform. If average power is being computed, a series of power spectrums is used. The average may be computed as a running average or a cumulative average.

The discrete power spectrum is computed similar to the Power Spectrum measurement *PWR_SPEC* with the **Spectrum Type** setting set to “Power Spectrum (Pwr/Bin)”.

The **Frequency Handling** setting is similar to PWR_SPEC’s **Frequencies Displayed** setting, and determines how negative frequencies are handled. If negative frequencies are folded, the channel frequency band will include both positive and negative frequency components. If negative frequencies are not folded, only the frequency components falling within the actual frequency band are included.

Channel power is then computed from the discrete power spectrum by summing the power values corresponding to the frequency bins contained within the channel frequency range. If a channel edge frequency is only partially contained within a bin, the power value corresponding to the bin is included if the frequency range extends over at least half the bin.

For example, suppose the measured signal has a center frequency of 10GHz and a sampling frequency of 10GHz. It contains three tones:

5 mW at 7.9 GHz

6 mW at 8.0 GHz

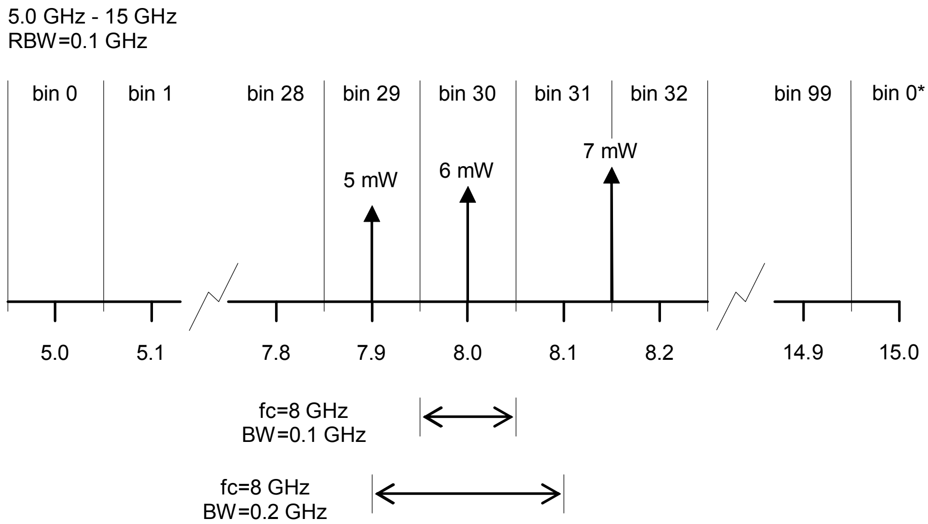
7 mW at 8.15 GHz

Assume that RBW has been set to 0.1 GHz. This results in an FFT containing 100 bins (10 GHz/0.1 GHz). Because the bins are always centered on the center frequency, the bin boundaries fall on the 0.05 GHz fraction, or ..., 7.85, 7.95, 8.05, ...

If the channel center frequency is set to 8 GHz and the channel bandwidth is set to 0.1 GHz, the channel band is 7.95 to 8.05 GHz, which is the single bin containing the 8 GHz tone. The measurement returns a channel power of 6 mW.

If the channel center frequency is set to 8 GHz and the channel bandwidth is set to 0.2 GHz, the channel band is 7.9 to 8.1 GHz, which is half of the bin containing the 7.9 GHz tone, the full bin containing the 8 GHz and half of the bin centered at 8.1

GHz. Because the frequency range covers at least half of the bin containing the 7.9 GHz tone, it's power contribution is included. The bin centered at 8.1 GHz is also covered by at least half the frequency range, so its power would be included. Due to the nature of the FFT, the tone at 8.15 GHz, which falls on a bin boundary, is entirely within the following bin, bin 32, and not in the preceding bin, bin 31. The measurement then returns a channel power of 11 mW.



Note that because the power measurements are performed using FFTs of the time-domain waveform, the simulation times when the measurements are generated are determined by several factors. The following determine how many data samples are required before a measurement can be computed.

If no averaging:

$$\#Samples_i = N \cdot i$$

If averaging:

$$\#Samples_i = N_{avg}N - (N_{avg} - 1)(1 - S_{ratio}) + S_{ratio}(i - 1)$$

where $\#Samples_i$ is the number of samples required for the i'th measurement, N_{avg} is the number of averages, and S_{ratio} is the sliding ratio between windows, where 1 represents a full data window width (0% overlap). The i'th measurement is output at simulation time $t_i = \#Samples_i / f_s$.

Meas. Filter and **Alpha** can be used to apply a measurement filter to the power spectrum prior to the computation of the channel power. The filter is applied in the frequency domain to the power spectrum after any averaging. The following filters are supported:

- Root-raised cosine:

$$H[f] = \begin{cases} 1 & \left(0 \le |f - f_{ch}| \le \frac{1-\alpha}{2T}\right) \\ \frac{1}{2}\left(1 + \cos\left(\frac{\pi T}{\alpha}\left(|f - f_{ch}| - \frac{1-\alpha}{2T}\right)\right)\right) & \left(\frac{1-\alpha}{2T} < |f - f_{ch}| \le \frac{1+\alpha}{2T}\right) \\ 0 & \left(\frac{1+\alpha}{2T} < |f - f_{ch}|\right) \end{cases}$$

where $T = 1/\text{Channel BW}$ and $f_{ch} = \text{Channel fc}$.

Cascaded Available Gain (Noise Operating Point): C_GA

Summary

C_GA computes cascaded available power gain using the VSS RF Budget Analysis simulator. The available power gain is the power gain that would be achieved if the output of a block were terminated in a matched load.

The measurement can display either overall cascaded gain versus frequency or the gain at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, the gain can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Linear Gain: C_GN

C_GN computes cascaded linear gain using the VSS RF Budget Analysis simulator. In nonlinear blocks with a gain parameter such as AMP_B, the linear gain is simply the gain parameter. Use the cascaded operating point gain measurement C_GP to measure the linearized operating point gain in nonlinear blocks.

The measurement can display either overall cascaded gain versus frequency or the gain at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, the gain can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Operating Point Gain: C_GP

C_GP computes cascaded operating point gain using the VSS RF Budget Analysis simulator. The operating point gain is based on the static signal power level of the input signal and typically represents the overall signal gain. Nonlinear blocks typically compute the operating point gain by passing a constellation of points representing a tone with the corresponding input signal power level through the nonlinearity and measuring the overall power gain.

The measurement can display either overall cascaded gain versus frequency or the gain at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, the gain can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Voltage Gain: C_GV

Summary

C_GV computes cascaded voltage gain using the VSS RF Budget Analysis simulator. The voltage gain is simply the total voltage at the end test point divided by the total voltage at the start test point. It is computed at the operating point, which is determined by the static signal power property. Nonlinear blocks typically compute the operating point gain by passing a constellation of points representing a tone with the corresponding input signal power level through the nonlinearity and measuring the overall power gain.

This measurement can display either overall cascaded voltage gain versus frequency, or the voltage gain at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed on the **Frequency Analysis** tab of the System Simulator Options dialog box. When displaying at specific frequencies, voltage gain can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

This measurement can be used with yield analysis and optimization.

This measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the block's PRIMINP parameter.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and RF Budget Analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Operating Point Headroom: C_HDRM

Summary

C_HDRM computes cascaded operating point headroom relative to P1dB using the VSS RF Budget Analysis simulator.

For blocks with a 1 dB compression point, the measurement displays the difference between the input referred 1 dB compression point of the block and the estimated signal power at the input to the block, when both values are in dBm. An optional margin can be specified that is subtracted from the 1 dB compression point:

$$HEADROOM_{dB} = IP1dB_{dBm} - MARGIN_{dB} - PIn_{dBm}$$

For other blocks, the measurement displays the difference between the input referred 1 dB compression point of the next downstream block that has a 1 dB compression point and the estimated signal power at the input to the block. If there is no downstream block with a 1 dB compression point then no value is generated.

The measurement can be configured to highlight on the system diagram any block that has a 1 dB compression point where the signal power at the input is greater than the 1 dB compression point less the margin.

C_HDRM can display either the headroom at the input of each block at a specific frequency offset, or the headroom at the input of the block prior to the end test point versus frequency. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog box. This measurement can be used with yield analysis and optimization.

C_HDRM requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the block's PRIMINP parameter.

The "*Visual System Simulator Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A

Name	Type	Range
End Test Point	System Test Point	N/A
Headroom Type	List of options	N/A
Margin in dB	Real value	Between -300 and 300
Highlight Negative Blocks	Checkbox	N/A
If using freqs for x-axis, display freqs	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Implementation Details

The measurement estimates the signal power using the propagated signal power property. You can view the value of this property using the Node Propagated Signal Power measurement *SPWR_node*.

Cascaded IP3: C_IP3

Summary

C_IP3 computes cascaded IP3 (3rd-order intercept point) using the VSS RF Budget Analysis simulator. See the "Computational Details" section for information on the computation of IP3.

This measurement can display either overall cascaded IP3 versus frequency, or the IP3 from at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog box. When displaying at specific frequencies, the IP3 can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

This measurement can be used with yield analysis and optimization.

C_IP3 requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
IP3 Type	List of options	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Noise Factor/Figure: C_NF

C_NF computes cascaded noise factor or figure using the VSS RF Budget Analysis simulator. The cascaded noise factor/figure is computed using the following:

$$F = \frac{T_{Out,\,Casc}}{T_{Start} \cdot G_{Av,\,Casc}}$$

where $T_{Out,Casc}$ is the overall noise temperature at the output of the block, $G_{Av,Casc}$ is the cascaded available power gain at the output of the block relative to the starting point, and T_{Start} is the noise temperature at the starting point. The noise temperatures can be measured using the Node noise temperature measurement ***T_node*** while the available power gain can be measured using the Cascaded available gain measurement ***C_GA***.

The measurement can display either overall cascaded noise factor/figure versus frequency or the noise factor/figure at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, the noise factor/figure can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A

Name	Type	Range
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

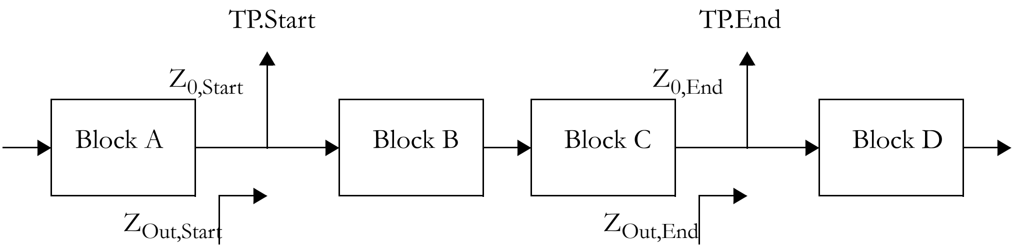
Cascaded S21: C_S21

Summary

C_S21 computes cascaded S21 using the VSS RF Budget Analysis simulator. S21 is computed as the reflected voltage at the end test point divided by the incident voltage at the start test point, or:

$$S21 = \frac{V^{-}_{End}}{V^{+}_{Start}} = \frac{V_{End}}{V_{Start}} \cdot \frac{\dfrac{Z_{Out,\,End} + Z_{0,\,End}}{2 \cdot Z_{Out,\,End}}}{\dfrac{Z_{Out,\,Start} + Z_{0,\,Start}}{2 \cdot Z_{Out,\,Start}}} \cdot \sqrt{\frac{Re\{Z_{0,\,Start}\}}{Re\{Z_{Out,\,End}\}}}$$

where the various impedances are:



In the above diagram, $Z_{0,Start}$ is the characteristic impedance of the output of Block A and $Z_{Out,Start}$ is the impedance seen looking out the output of Block A. $Z_{0,End}$ is the characteristic impedance of the output of Block C and $Z_{Out,End}$ is the impedance seen looking out the output of Block C.

The voltage gain V_{End}/V_{Start} is similar to the voltage gain measured by the Cascaded Voltage Gain measurement C_GV.

NOTE. Impedance mismatch modeling should be enabled when using this measurement. If it is not enabled, the results will be identical to the Cascaded Voltage Gain measurement, as the impedances will all be identical.

The measurement can display either overall cascaded S21 versus frequency or the S21 at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, S21 can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Signal To Noise Ratio: C_SNR

Summary

C_SNR computes an estimate of the cascaded signal-to-noise ratio using the VSS RF Budget Analysis simulator. The signal power is determined from the static signal power property. The noise power is estimated from the cascaded noise PSD levels within the specified noise bandwidth. The signal is always assumed to be centered at the center frequency.

The measurement supports the specification of a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
Signal Type	List of options	N/A
Noise Type	List of options	N/A
Noise Bandwidth	Real value	Varies

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

C_SNR uses the static signal power property for the signal power level. The static signal power property can be viewed using the Node Propagated Signal Power measurement *SPWR_node*, the Static Signal Power Level annotation *SIGPWR* or the Static Signal Power Properties measurement *PWR_PROP*.

The noise power is computed from the noise PSD values within the specified noise bandwidth. To simplify the computations, the measurement assumes the noise PSD varies linearly between specified frequency points. The noise power between two adjacent frequency points f_a and f_b with noise PSD of $N0_a$ and $N0_b$ is then:

$$N_{ab} = |f_b - f_a| \cdot (N0_a + N0_b)/2$$

The source of the noise PSD is determined by the **Noise Type** setting. The following choices are available:

Circuit and channel: The noise PSD includes circuit noise such as that corresponding to noise figure, and channel noise such as that generated by the AWGN channel block.

Circuit only: The noise PSD only include circuit noise.

The noise PSD for either type can be measured separately using the Node noise PSD measurement *N0_node*.

The noise frequency band is determined by the **Noise Bandwidth** setting and whether the signal is a complex or real signal. If the **Noise Bandwidth** setting is set to "Data Rate", the bandwidth is determined by dividing the sampling frequency of the signal by the oversampling rate (samples per symbol).

If the noise bandwidth is f_{BW}, the noise frequency band is then:

$$max\left(0, f_c - \frac{f_{BW}}{2}\right) \le f_N \le f_c + \frac{f_{BW}}{2}$$

For complex signals f_c is the center frequency associated with the signal. For real signals f_c is 0.

Cascaded Equivalent Input Noise Temperature: C_TE

Summary

C_TE computes the cascaded equivalent input noise temperature using the VSS RF Budget Analysis simulator. For a given segment of a signal path, if the input noise temperature is T_{In}, the output noise temperature is T_{Out}, and the available power gain is G_{Av}, the equivalent input noise temperature T_E is given by:

$$T_E = \frac{T_{Out}}{G_{Av}} - T_{In}$$

The input and output noise temperatures can be viewed separately using the Node noise temperature measurement ***T_node***. The available power gain can be viewed using the Cascaded available gain measurement ***C_GA***.

The measurement can display either overall cascaded T_E versus frequency or T_E at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, T_E can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Cascaded Equivalent Output Noise Temperature: C_TN

C_TN computes the cascaded equivalent output noise temperature using the VSS RF Budget Analysis simulator. For a given segment of a signal path, if the input noise temperature is T_{In}, the output noise temperature is T_{Out}, and the available power gain is G_{Av}, the equivalent output noise temperature T_N is given by:

$$T_E = T_{Out} - T_{In} \cdot G_{Av}$$

The input and output noise temperatures can be viewed separately using the Node noise temperature measurement ***T_node***. The available power gain can be viewed using the Cascaded available gain measurement ***C_GA***.

The measurement can display either overall cascaded T_N versus frequency or T_N at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. When displaying at specific frequencies, T_N can be displayed as either cascaded from the starting point to the output of the block, or as the contribution of each block.

The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Block Output	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Node Noise PSD (N0): N0_node

N0_node displays noise PSD computed using the VSS RF Budget Analysis simulator. The noise PSD can be restricted to only circuit noise generated by RF blocks or can include both circuit and channel noise.

The measurement can display either overall output noise PSD versus frequency or the noise PSD at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end test points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
Noise Type	List of options	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value in units of power/Hz.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Node Propagated Signal Power: SPWR_node

SPWR_node displays the propagated signal power property at a node using the VSS RF Budget Analysis simulator. The propagated signal power property is used by several RF Budget Analysis measurements, including Cascaded Operating Point Gain, C_GP, and Cascaded Operating Point Headroom, C_HDRM.

This measurement can display either overall signal power versus frequency or the signal power at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog box. The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end test points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the block's PRIMINP parameter.

The "*Visual System Simulator Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value in units of power.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Node Noise Temperature: T_node

T_node displays noise temperature computed using the VSS RF Budget Analysis simulator. The noise temperature can be restricted to only circuit noise generated by RF blocks or can include both circuit and channel noise.

The measurement can display either overall output noise temperature versus frequency or the noise temperature at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end test points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
Noise Type	List of options	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value in temperature units.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Node Noise Voltage: Vn_node

Vn_node displays the voltage spectral density of thermal noise at a node that is delivered to the load. Vn_node uses the VSS RF Budget Analysis simulator.

The measurement can display either overall $V/\sqrt{Hz}$ versus frequency or $V/\sqrt{Hz}$ at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog boxes. The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end test points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
Noise Type	List of options	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a real value in temperature units.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Node Impedance: Z_node

Z_node displays the impedance seen looking into or out of output nodes. Z_node uses the VSS RF Budget Analysis simulator.

This measurement can display either overall impedance versus frequency or the impedance at the output of each block at specific frequencies. The frequencies are determined from the center frequency of the signal and the frequency offsets listed under the **Frequency Analysis** tab of the System Simulator Options dialog box. The measurement can be used with yield analysis and optimization.

The measurement requires a start and an end test point on a primary signal path, with the data flowing from the start to the end. It also allows a single point where the start and end test points are the same. Note that if the signal passes through a block that combines multiple input signals such as a combiner or adder, only one of the input signals to that path can be configured as the primary signal path. This is done through the PRIMINP parameter of the block.

The "*VSS Modeling Guide*" contains more information on VSS frequency domain analysis and the RF budget analysis measurements.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Start Test Point	System Test Point	N/A
End Test Point	System Test Point	N/A
Direction	List of options	N/A
If using freqs for x-axis, display freqs	List of options	N/A
Sweep Freq (FDOC)	List of options	N/A

Result

This measurement returns a complex value in units of resistance.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

RFI Current Spectrum: RFI_I_SPEC

Summary

RFI_I_SPEC computes a current spectrum using RF Inspector, allowing you to view the various contributors to individual frequency components of the spectrum.

Clicking and holding the cursor on a frequency component of the measurement's spectrum curve in the graph window displays the top two contributors to the selected component. Double-clicking on a frequency component opens the inspection window, allowing you to trace the individual contributors of the frequency component.

NOTE. RF Inspection is normally only available with the VSS RF blocks. These include most of the blocks in the RF Blocks category and the circuit based filters in the Filters category (Bandpass, Bandstop, Lowpass and Highpass sub-categories).

The I/Q and OFDM modulators, as well as the transmitters based on the modulators also support RF Inspection. This includes transmitters such as BPSK, MPSK, QAM, and QPSK.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point To Analyze	System Meter	N/A
Test Point Identifying Signal	System Meter	N/A
Components to Display	List of Options	N/A
Frequency Range to Display	List of Options	N/A
Lower Frequency	Real value	>= 0
Upper Frequency	Real value	>= 0
Data Output	List of Options	N/A

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *Visual System Simulator Modeling Guide* for details.

Result

This measurement returns the RF Inspector current spectrum at a selected test point, a measurement probe, or a port of an RF block.

You can display the values in dB by selecting the **dB** checkbox in the Add/Modify Measurement dialog box.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid. The RF Inspector window is only available when the measurement is displayed on a rectangular graph.

Computational Details

The RF Inspector (or RFI) simulator is a VSS frequency domain circuit simulator that tracks the various contributors to frequency components at ports, to blocks that support RF Inspection.

The RFI simulator computes the voltage at each RF node, and the current flowing into each RF port. Positive values for current indicate current flowing into the RF port, and negative values indicate current flowing out of the RF port.

You can use the **Test Point To Analyze** and **Test Point Identifying Signal** settings with VSS meter blocks such as TP and VSA, the measurement probe M_PROBE, and individual ports on RF blocks. When used with the VSS meter blocks, the measurement monitors the spectrum at the output port of the block that generates the signal measured by the meter.

Using an M_PROBE allows you to quickly monitor the spectrum at any port. You can drag the M_PROBE from port to port, and the measurement updates immediately to reflect the spectrum at the new port.

This measurement classifies frequency components and their contributions into several categories:

- Signal - The signal of interest, as identified by the **Test Point Identifying Signal** setting.
- Distortion - Any contribution that is dependent on Signal, but is not the Signal.
- Interference - Any contribution that is neither a Signal nor a Distortion. If the Signal is turned off, what remains is Interference.

The RF Inspector window tags each contributor with an icon indicating the contributor's category.

The **Components to Display** setting allows you to select individual categories for display.

The **Data Output** setting is used to configure the measurement data for use with output equations. The data points generated by the measurement may include dummy values to help in displaying the spectrum in a graph, particularly when working with modulated signals. These data points have values set to approximately 1.798e308 and serve as markers indicating the edges of frequency components.

If the **Data Output** setting is set to "Data Only", the measurement does not include any dummy values in the output.

MODULATED SIGNALS

When working with supported modulated signals, the RFI simulator initially approximates the modulated signal with a frequency component whose total power and bandwidth are similar to the modulated signal. The power is distributed within the frequency component to roughly match the shape of the modulated signal. If the frequency component then passes through a nonlinearity, the generated IM products will have the appropriate total power and bandwidth, but will not retain the approximate shape of the modulated signal.

IMPEDANCE MISMATCH

Because the RFI simulator is a circuit simulator, it always generates results with impedance mismatch effects. However, the impedance mismatch effects setting in the System Simulator Options dialog box may still affect the simulation results.

If impedance mismatch effects are not enabled, the RFI simulator automatically terminates any unterminated ports on RF blocks with a load of the default characteristic impedance from the System Simulator Options dialog box.

If impedance mismatch effects are enabled, unterminated ports on RF blocks remain unterminated.

RFI Power in Frequency Band: RFI_PWR_BAND

Summary

RFI_PWR_BAND computes the power within a specified frequency band using RF Inspector. RF Inspector allows you to view the various contributors to individual frequency components of the spectrum.

Clicking and holding the cursor on a frequency component of the measurement's spectrum curve in the graph window displays the top two contributors to the selected component. Double-clicking on a frequency component opens the inspection window, allowing you to trace the individual contributors of the frequency component.

NOTE. RF Inspection is normally only available with the VSS RF blocks. These include most of the blocks in the RF Blocks category and the circuit based filters in the Filters category (Bandpass, Bandstop, Lowpass and Highpass subcategories).

The I/Q and OFDM modulators, as well as the transmitters based on the modulators also support RF Inspection. This includes transmitters such as BPSK, MPSK, QAM, and QPSK.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point To Analyze	System Meter	N/A
Test Point Identifying Signal	System Meter	N/A
Components to Display	List of Options	N/A
Frequency Range to Display	List of Options	N/A
Lower Frequency	Real value	>= 0
Upper Frequency	Real value	>= 0

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *Visual System Simulator Modeling Guide* for details.

Result

This measurement returns the RF Inspector frequency band power at a selected test point, a measurement probe, or a port of an RF block.

You can display the values in dB by selecting the **dB** checkbox in the Add/Modify Measurement dialog box.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid. The RF Inspector window is only available when the measurement is displayed on a rectangular graph.

Computational Details

The RF Inspector (or RFI) simulator is a VSS frequency domain circuit simulator that tracks the various contributors to frequency components at ports, to blocks that support RF Inspection.

The RFI simulator computes the voltage at each RF node, and the current flowing into each RF port. Positive values for current indicate current flowing into the RF port, and negative values indicate current flowing out of the RF port.

The **Test Point To Analyze** and **Test Point Identifying Signal** settings can be used with VSS meter blocks such as TP and VSA, the measurement probe M_PROBE, and individual ports on RF blocks. When used with the VSS meter blocks, the measurement monitors the spectrum at the output port of the block that generates the signal measured by the meter.

Using an M_PROBE allows you to quickly monitor the spectrum at any port. You can drag the M_PROBE from port to port and the measurement will update immediately to reflect the spectrum at the new port.

This measurement classifies frequency components and their contributions into several categories:

- Signal - The signal of interest, as identified by the **Test Point Identifying Signal** setting.
- Distortion - Any contribution that is dependent on Signal, but is not the Signal.
- Interference - Any contribution that is neither a Signal nor a Distortion. If the Signal were turned off, what remains is Interference.

The RF Inspector window tags each contributor with an icon indicating the contributor's category.

The **Components to Display** setting allows you to select a specific category to be measured.

The **Lower Frequency** and **Upper Frequency** settings specify the edges of the frequency band to be measured. The frequencies can be specified as either absolute frequency values or relative to the center frequency at the measurement point. Specifying relative frequencies is useful when working with swept frequencies or with mixers.

MODULATED SIGNALS

When working with supported modulated signals, the RFI simulator initially approximates the modulated signal with a frequency component whose total power and bandwidth are similar to the modulated signal. The power is distributed within the frequency component to roughly match the shape of the modulated signal. If the frequency component then passes through a nonlinearity, the generated IM products have the appropriate total power and bandwidth, but do not retain the approximate shape of the modulated signal.

IMPEDANCE MISMATCH

Because the RFI simulator is a circuit simulator, it always generates results with impedance mismatch effects. However, the impedance mismatch effects setting in the System Simulator Options dialog box may still affect the simulation results.

If impedance mismatch effects are not enabled, the RFI simulator automatically terminates any unterminated ports on RF blocks with a load of the default characteristic impedance from the System Simulator Options dialog box.

If impedance mismatch effects are enabled, unterminated ports on RF blocks remain unterminated.

RFI Power Spectrum: RFI_PWR_SPEC

Summary

RFI_PWR_SPEC computes a power spectrum using RF Inspector, allowing you to view the various contributors to individual frequency components of the spectrum.

Clicking and holding the cursor on a frequency component of the measurement's spectrum curve in the graph window displays the top two contributors to the selected component. Double-clicking a frequency component opens the inspection window, allowing you to trace the individual contributors of the frequency component.

NOTE. RF Inspection is normally only available with the VSS RF blocks. These include most of the blocks in the RF Blocks category and the circuit based filters in the Filters category (Bandpass, Bandstop, Lowpass and Highpass sub-categories).

The I/Q and OFDM modulators, as well as the transmitters based on the modulators, also support RF Inspection. This includes transmitters such as BPSK, MPSK, QAM, and QPSK.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point To Analyze	System Meter	N/A
Test Point Identifying Signal	System Meter	N/A
Components to Display	List of Options	N/A
Frequency Range to Display	List of Options	N/A
Lower Frequency	Real value	>= 0
Upper Frequency	Real value	>= 0
Data Output	List of Options	N/A

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *Visual System Simulator Modeling Guide* for details.

Result

This measurement returns the RF Inspector power spectrum at a selected test point, a measurement probe, or a port of an RF block.

You can display the values in dB by selecting the **dB** checkbox in the Add/Modify Measurement dialog box.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid. The RF Inspector window is only available when the measurement is displayed on a rectangular graph.

Computational Details

The RF Inspector (or RFI) simulator is a VSS frequency domain circuit simulator that tracks the various contributors to frequency components at ports, to blocks that support RF Inspection.

The RFI simulator computes the voltage at each RF node, and the current flowing into each RF port. Positive values for current indicate current flowing into the RF port and negative values indicate current flowing out of the RF port.

The **Test Point To Analyze** and **Test Point Identifying Signal** settings can be used with VSS meter blocks such as TP and VSA, the measurement probe M_PROBE, and individual ports on RF blocks. When used with the VSS meter blocks, the measurement monitors the spectrum at the output port of the block that generates the signal measured by the meter.

Using an M_PROBE allows you to quickly monitor the spectrum at any port. You can drag the M_PROBE from port to port and the measurement updates immediately to reflect the spectrum at the new port.

The measurement classifies frequency components and their contributions into several categories:

- Signal - The signal of interest, as identified by the **Test Point Identifying Signal** setting.
- Distortion - Any contribution that is dependent on Signal, but is not the Signal.
- Interference - Any contribution that is neither a Signal nor a Distortion. If the Signal were turned off, what remains is Interference.

The RF Inspector window tags each contributor with an icon indicating the contributor's category.

The **Components to Display** setting allows you to select individual categories for display.

The **Data Output** setting is used to configure the measurement data for use with output equations. The data points generated by the measurement may include dummy values to help in displaying the spectrum in a graph, particularly when working with modulated signals. These data points have values set to approximately 1.798e308 and serve as markers indicating the edges of frequency components.

If the **Data Output** setting is set to **Data Only**, the measurement does not include any dummy values in the output.

MODULATED SIGNALS

When working with supported modulated signals, the RFI simulator initially approximates the modulated signal with a frequency component whose total power and bandwidth are similar to the modulated signal. The power is distributed within the frequency component to roughly match the shape of the modulated signal. If the frequency component then passes through a nonlinearity, the generated IM products have the appropriate total power and bandwidth, but do not retain the approximate shape of the modulated signal.

IMPEDANCE MISMATCH

Because the RFI simulator is a circuit simulator, it always generates results with impedance mismatch effects. However, the impedance mismatch effects setting in the System Simulator Options dialog box may still affect the simulation results.

If impedance mismatch effects are not enabled, the RFI simulator automatically terminates any unterminated ports on RF blocks with a load of the default characteristic impedance from the System Simulator Options dialog box.

If impedance mismatch effects are enabled, unterminated ports on RF blocks remain unterminated.

RFI Voltage Spectrum: RFI_V_SPEC

Summary

RFI_V_SPEC computes a voltage spectrum using RF Inspector, allowing you to view the various contributors to individual frequency components of the spectrum.

Clicking and holding the cursor on a frequency component of the measurement's spectrum curve in the graph window displays the top two contributors to the selected component. Double-clicking a frequency component opens the inspection window, allowing you to trace the individual contributors of the frequency component.

NOTE. RF Inspection is normally only available with the VSS RF blocks. These include most of the blocks in the RF Blocks category and the circuit based filters in the Filters category (Bandpass, Bandstop, Lowpass and Highpass sub-categories).

The I/Q and OFDM modulators, as well as the transmitters based on the modulators also support RF Inspection. This includes transmitters such as BPSK, MPSK, QAM, and QPSK.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point To Analyze	System Meter	N/A
Test Point Identifying Signal	System Meter	N/A
Components to Display	List of Options	N/A
Frequency Range to Display	List of Options	N/A
Lower Frequency	Real value	>= 0
Upper Frequency	Real value	>= 0
Data Output	List of Options	N/A

NOTE. If the selected system diagram is configured for a swept simulation, the measurement has additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "Simulation Basics" chapter of the *Visual System Simulator Modeling Guide* for details.

Result

This measurement returns the RF Inspector voltage spectrum at a selected test point, a measurement probe, or a port of an RF block.

You can display the values in dB by selecting the **dB** checkbox in the Add/Modify Measurement dialog box.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid. The RF Inspector window is only available when the measurement is displayed on a rectangular graph.

Computational Details

The RF Inspector (or RFI) simulator is a VSS frequency domain circuit simulator that tracks the various contributors to frequency components at ports, to blocks that support RF Inspection.

The RFI simulator computes the voltage at each RF node and the current flowing into each RF port. Positive values for current indicate current flowing into the RF port, and negative values indicate current flowing out of the RF port.

The **Test Point To Analyze** and **Test Point Identifying Signal** settings can be used with VSS meter blocks such as TP and VSA, the measurement probe M_PROBE, and individual ports on RF blocks. When used with the VSS meter blocks, the measurement monitors the spectrum at the output port of the block that generates the signal measured by the meter.

Using an M_PROBE allows you to quickly monitor the spectrum at any port. You can drag the M_PROBE from port to port, and the measurement updates immediately to reflect the spectrum at the new port.

This measurement classifies frequency components and their contributions into several categories:

- Signal - The signal of interest, as identified by the **Test Point Identifying Signal** setting.
- Distortion - Any contribution that is dependent on Signal, but is not the Signal.
- Interference - Any contribution that is neither a Signal nor a Distortion. If the Signal were turned off, what remains is Interference.

The RF Inspector window tags each contributor with an icon indicating the contributor's category.

The **Components to Display** setting allows you to select individual categories for display.

The **Data Output** setting is used to configure the measurement data for use with output equations. The data points generated by the measurement may include dummy values to help in displaying the spectrum in a graph, particularly when working with modulated signals. These data points have values set to approximately 1.798e308 and serve as markers indicating the edges of frequency components.

If the **Data Output** setting is set to **Data Only**, the measurement does not include any dummy values in the output.

MODULATED SIGNALS

When working with supported modulated signals, the RFI simulator initially approximates the modulated signal with a frequency component whose total power and bandwidth are similar to the modulated signal. The power is distributed within the frequency component to roughly match the shape of the modulated signal. If the frequency component then passes through a nonlinearity, the generated IM products have the appropriate total power and bandwidth, but do not retain the approximate shape of the modulated signal.

IMPEDANCE MISMATCH

Because the RFI simulator is a circuit simulator, it always generates results with impedance mismatch effects. However, the impedance mismatch effects setting in the System Simulator Options dialog box may still affect the simulation results.

If impedance mismatch effects are not enabled, the RFI simulator automatically terminates any unterminated ports on RF blocks with a load of the default characteristic impedance from the System Simulator Options dialog box.

If impedance mismatch effects are enabled, unterminated ports on RF blocks remain unterminated.

Power Spectrum: PWR_SPEC

Summary

PWR_SPEC computes the power spectrum of a time domain signal. The spectrum can be configured to display power spectral density (Power/Hz) or power spectrum (Power/Frequency Point). For a normalized power spectrum use PWR_SPECN.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
Frequency Axis Scaling	List of options	N/A
Frequencies Displayed	List of options	N/A
Y-Axis Output	List of options	N/A
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value in power units. The power can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The power spectrum measurement is based on performing fast Fourier transforms (FFT) on the time domain signal. The number of bins in the FFT is determined by the **RBW/#FFT Bins** setting based on the units of the setting:

Auto: The number of FFT bins is determined from the test point.

 # FFT Bins: The value is the number of FFT bins.

Hz, kHz, MHz, GHz: The setting is resolution bandwidth. The number of FFT bins is computed using:

$$N_{FFT} = ceil(f_s/f_{RBW})$$

where f_s is the sampling frequency, f_{RBW} is the resolution bandwidth and $ceil(x)$ returns the smallest whole number greater than or equal to x.

The frequency resolution of the spectrum is then f_s/N_{FFT}.

The **VBW/#Avg.** setting controls the smoothness of the spectrum through the use of averaging. The averaging details are explained later. The number of averages computed from the **VBW/#Avg.** setting based on the units of the setting:

Auto: The number of averages is determined from the test point.

 # Averages: The value is the number of averages. The value may be set to 0 to indicate cumulative averaging.

Cumulative: Cumulative averaging is performed from T=0. The setting's value is ignored.

Hz, kHz, MHz, GHz: The setting is treated as an approximate video bandwidth. This value is converted to a number of averages using:

$$N_{Avg} = ceil(f_s/N_{FFT}/f_{VBW})$$

where f_{VBW} is the video bandwidth. This approximation is based solely on the observation that reducing the video bandwidth on a spectrum analyzer smooths the spectrum display. The smoothness derived from N_{Avg} and the smoothness in a spectrum analyzer with the same resolution bandwidth setting will not necessarily be equivalent.

When $N_{Avg} > 1$ the **Overlap Ratio for Averaging** setting determines the amount each FFT is advanced when computing the average. The setting is the fraction of a full FFT width of samples to advanced between successive FFTs. A value ≤ 0 results in the overlap being determined from the test point.

The **FFT Windowing Type** and **FFT Windowing Parameter** settings are used to apply a window to the time domain signal prior to computing the FFT. Windowing helps reduce sidelobes introduced by the FFT process when the signal is not periodic relative to the FFT width. When the signal is periodic, windowing will spread out the spectrum rather than displaying single tones.

If **FFT Windowing Type** is set to "Auto", the windowing will be determined by the test point. By default no windowing will be applied if NAvg = 1. If **FFT Windowing Type** is not "Auto", the windowing will be performed regardless of whether averaging is being performed. ("Rectangular" is the same as no windowing.)

The other windowing options are the same as those found in the Test Point block TP. Refer to TP for more details.

The **Frequency Axis Scaling** setting is used to select how the frequency axis is displayed:

Auto: The setting will be determined by the test point.

Absolute: The frequencies will be absolute frequencies, ranging from $f_c - f_s/2$ to $f_c + f_s/2$.

Normalized to +/- 0.5: The frequencies will be normalized to the sampling frequency and range from -0.5 to 0.5.

Normalized to +/- Ns/2: The frequencies will be normalized to the symbol rate.

Relative to center frequency: The frequencies will be relative to the center frequency and range from $-f_s/2$ to $+f_s/2$.

FFT Bin number: The x-axis will display the frequency bin number, with the bin corresponding to the smallest frequency bin 0.

The **Frequencies Displayed** setting determines how negative absolute frequency values are treated:

Spectrum Analyzer Style: Negative frequencies are treated similar to a spectrum analyzer. For real signals and complex envelope signals with non-zero center frequency negative frequencies are 'folded' onto the corresponding positive frequencies as in the "Non-negative, folded" option. For complex signals with zero center frequency (baseband signals) both positive and negative frequencies are displayed.

All: All frequencies are displayed.

Non-negative, truncated: Negative frequencies are ignored. If there is significant power in the negative frequencies, a warning message will be displayed. Care should be used when using this option.

Non-negative, folded: Negative frequencies are 'folded' onto the corresponding positive frequencies by adding the complex conjugate of the negative frequency component to the corresponding positive frequency component. This results in 'real frequency' power values.

The power spectrum is computed differently depending on whether frequency folding is being performed. When no folding is performed, the power spectrum is computed based on the following modified periodogram:

$$PS_{xx}[f_k] = \frac{\frac{1}{N_{FFT}} \sum_{i=0}^{N_{FFT}-1} \left|w_i x_i e^{-j2\pi ik/N_{FFT}}\right|^2}{\sum_{i=0}^{N_{FFT}-1} \left|w_i\right|^2}, \qquad k = 0, 1, \ldots, N_{FFT}-1$$

where $PS_{xx}[f_k]$ is the power contained in the frequencies between $(f_{k-1} + f_k) / 2$ and $(f_k + f_{k+1}) / 2$, x_i is the sequence of measured values and w_i is the optional FFT windowing function.

The power spectrum values represented by $PS_{xx}[f_k]$ may be directly summed to obtain the total power within the frequency range. This is the computation performed by the Power Meter (PWR_MTR) and Power vs. Time (PWR_vsT) measurement.

When averaging is being performed, the average power spectrum is computed by averaging several periodograms:

$$PS[f_k]_{Avg} = \frac{1}{N_{Avg}} \cdot \sum_{N_{Avg}} PS_{xx}[f_k]_i$$

where each subsequent periodogram is advanced by Overlap Ratio $\cdot$ N_{FFT} samples from the previous periodogram.

When frequency folding is being performed, the voltage spectrum is first computed using the modified discrete Fourier transform:

$$V[f_k] = \frac{\sum_{i=0}^{N_{FFT}-1} w_i x_i e^{-j2\pi ik/N_{FFT}}}{\sum_{i=0}^{N_{FFT}-1} w_i}, \qquad k = 0, 1, \ldots, N_{FFT}-1$$

where $V[f_k]$ is the amplitude of frequency f_k, x_i is the sequence of measured values and w_i is the optional FFT windowing function. For $w_i = 1$, $i = 0, 1, ..., N$ -1 (no windowing), the voltage spectrum equation is the normal discrete Fourier transform.

The folding is accomplished by adding the complex conjugate of the negative frequency term to the corresponding positive frequency bin. If the negative frequency terms do not correspond exactly to a positive frequency term, the conjugate term is added proportionately to the two positive frequency bins closest to the desired frequency using:

$$V'[f_k] = V[f_k] + V^*[-f_l] \cdot (-f_l - f_k)/(f_{k+1} - f_k)$$
$$V'[f_{k+1}] = V[f_{k+1}] + V^*[-f_l] \cdot (f_{k+1} + f_l)/(f_{k+1} - f_k)$$

where f_l is the negative frequency and f_k, f_{k+1} are the two positive frequency bins closest to $-f_l$, and $f_k < -f_l < f_{k+1}$.

The power spectrum is then computed from the folded voltage spectrum using:

$$PS_{xx}[f_k] = \frac{1}{2}\left|V'[f_k]\right|^2$$

where f_k ranges from the larger of 0 or $f_c - f_s/2$ to $f_c + f_s/2$.

When averaging is performed, the average power spectrum is computed by averaging the individual power spectrums.

For both the folded and non-folded cases, if the signal is a complex envelope signal with non-zero center frequency, the DC power component, if any, is adjusted by a factor of 2 to convert the RMS power value $PS_{xx}[0]$ to DC power.

The **Y-Axis Output** setting determines the final scaling of the power spectrum:

Power Spectrum (Pwr/Bin): $PS_{xx}[f_k]$ or $PS_{xx}[f_k]_{Avg}$ is displayed and represents the power in each frequency bin. The sum of the values is the average power in the frequency band.

PSD (Pwr/Hz): The power spectral density, in power per Hz, is displayed. The PSD at frequency f_k is $PS_{xx}[f_k] * N_{FFT}/f_s$.

Spectrum Analyzer Spectrum: Windowing similar to a spectrum analyzer is applied. The value read on the y-axis corresponds to the frequency content near the frequency bin even when spectral spreading occurs. Computing the power within a frequency band requires adjustment by the noise power bandwidth of the window function.

The **Peak Hold, Power Spectrum** and **Peak Hold, PSD**, and **Peak Hold, Analyzer Spectrum** options display peak y-axis values for the corresponding scaling. The **Start Offset setting** can be used to allow the signal to settle before starting the peak capture. Note that peak hold is not currently supported when cumulative averaging is enabled.

Power Spectrum, Normalized: PWR_SPECN

Summary

PWR_SPECN computes the normalized power spectrum of a time domain signal. The spectrum can be configured to display normalized power spectral density (Power/Hz) or normalized power spectrum (Power/Frequency Point).

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Avg.	Real value	>0
Frequency Axis Scaling	List of options	N/A
Frequencies Displayed	List of options	N/A
Y-Axis Output	List of options	N/A
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value in power units. The power can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

PWR_SPECN is similar to the Power Spectrum measurement *PWR_SPEC* except the y-axis values are normalized. If the measurement is set to display in dB, the y-axis values are offset so the maximum value is 0 dB. If the measurement is not set to display in dB, the y-axis values are offset and scaled so the smallest power value is set to 0 and the largest power value is set to 1.

Plot Spectral Mask from File: SPECMASK

Summary

SPECMASK plots a spectral mask specified in a Text Data File object under the **Data Files** node of the Project Browser. You specify the column representing the x-axis (frequency) and the column representing the y-axis (spectral values of the mask, usually in dB).

Parameters

Name	Type	Range
Data File Name	Subcircuit	Text file contain two or more columns
Column for X Axis	Integer	1 to 1000
Column for X Axis	Integer	1 to 1000

Result

SPECMASK returns a real value, the spectral mask. You can also display the value can in dB by selecting the **DB** check box under **Result Type**, but that makes sense only when the spectral mask values are not specified in dB themselves, which is a rather unusual situation.

Graph Type

SPECMASK displays on a Rectangular graph or Tabular grid.

Computational Details

Although the measurement is as flexible as the PlotCol measurement (under Data), its most usual usage would involve a text file, the first column of which lists the frequency values, and the second column lists the values of the spectral mask. The columns must be tab-separated. An exclamation point at the beginning of a line indicates a comment line. An example of a data file follows, containing the spectral mask for a signal according to the IEEE 802.11b standard, centered at 2.437 GHz:

Freq(,MHz) (Mag,dB)

2404 -50

2415 -50

2415 -30

2426	-30
2426	0
2437	0
2448	0
2448	-30
2459	-30
2459	-50
2470	-50

! This is the end of the file

Voltage Spectrum: V_SPEC

Summary

V_SPEC computes the voltage spectrum of a time domain signal.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
RBW / # FFT Bins	Real value	>0
VBW / # Averages	Real value	>0
Frequency Axis Scaling	List of options	N/A
Frequencies Displayed	List of options	N/A
Spectrum Type	List of options	N/A
*Overlap Ratio for Averaging	Real value	<=1
*FFT Windowing Type	List of options	N/A
*FFT Windowing Parameter	Real value	Varies
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns a real value in voltage units. The voltage can be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units.

Right-clicking the cursor on the measurement when it is displayed on a rectangular graph will display the settings used to compute the result at the selected data point.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The voltage spectrum measurement is based on performing fast Fourier transforms (FFT) on the time domain signal. The number of bins in the FFT is determined by the **RBW/#FFT Bins** setting based on the units of the setting:

Auto: The number of FFT bins is determined from the test point.

 # FFT Bins: The value is the number of FFT bins.

Hz, kHz, MHz, GHz: The setting is resolution bandwidth. The number of FFT bins is computed using:

$$N_{FFT} = ceil(f_s/f_{RBW})$$

where f_s is the sampling frequency, f_{RBW} is the resolution bandwidth and $ceil(x)$ returns the smallest whole number greater than or equal to x.

The frequency resolution of the spectrum is then f_s/N_{FFT}.

The **VBW/#Avg.** setting controls the smoothness of the spectrum through the use of averaging. The averaging details are explained later. The number of averages computed from the **VBW/#Avg.** setting based on the units of the setting:

Auto: The number of averages is determined from the test point.

 # Averages: The value is the number of averages. The value may be set to 0 to indicate cumulative averaging.

Cumulative: Cumulative averaging is performed from T=0. The setting's value is ignored.

Hz, kHz, MHz, GHz: The setting is treated as an approximate video bandwidth. This value is converted to a number of averages using:

$$N_{Avg} = ceil(f_s/N_{FFT}/f_{VBW})$$

where f_{VBW} is the video bandwidth. This approximation is based solely on the observation that reducing the video bandwidth on a spectrum analyzer smooths the spectrum display. The smoothness derived from N_{Avg} and the smoothness in a spectrum analyzer with the same resolution bandwidth setting will not necessarily be equivalent.

When $N_{Avg} > 1$ the **Overlap Ratio for Averaging** setting determines the amount each FFT is advanced when computing the average. The setting is the fraction of a full FFT width of samples to advanced between successive FFTs. A value ≤ 0 results in the overlap being determined from the test point.

The **FFT Windowing Type** and **FFT Windowing Parameter** settings are used to apply a window to the time domain signal prior to computing the FFT. Windowing helps reduce sidelobes introduced by the FFT process when the signal is not periodic relative to the FFT width. When the signal is periodic, windowing will spread out the spectrum rather than displaying single tones.

If **FFT Windowing Type** is set to "Auto", the windowing will be determined by the test point. By default no windowing will be applied if NAvg = 1. If **FFT Windowing Type** is not "Auto", the windowing will be performed regardless of whether averaging is being performed. ("Rectangular" is the same as no windowing.)

The other windowing options are the same as those found in the Test Point block TP. Refer to TP for more details.

The **Frequency Axis Scaling** setting is used to select how the frequency axis is displayed:

Auto: The setting will be determined by the test point.

Absolute: The frequencies will be absolute frequencies, ranging from $f_c - f_s/2$ to $f_c + f_s/2$.

Normalized to +/- 0.5: The frequencies will be normalized to the sampling frequency and range from -0.5 to 0.5.

Normalized to +/- Ns/2: The frequencies will be normalized to the symbol rate.

Relative to center frequency: The frequencies will be relative to the center frequency and range from $-f_s/2$ to $+f_s/2$.

FFT Bin number: The x-axis will display the frequency bin number, with the bin corresponding to the smallest frequency bin 0.

The **Frequencies Displayed** setting determines how negative absolute frequency values are treated:

Spectrum Analyzer Style: Negative frequencies are treated similar to a spectrum analyzer. For real signals and complex envelope signals with non-zero center frequency negative frequencies are 'folded' onto the corresponding positive frequencies as in the "Non-negative, folded" option. For complex signals with zero center frequency (baseband signals) both positive and negative frequencies are displayed.

All: All frequencies are displayed.

Non-negative, truncated: Negative frequencies are ignored. If there is significant power in the negative frequencies, a warning message will be displayed. Care should be used when using this option.

Non-negative, folded: Negative frequencies are 'folded' onto the corresponding positive frequencies by adding the complex conjugate of the negative frequency component to the corresponding positive frequency component.

The voltage spectrum is computed based on the following modified discrete Fourier transform:

$$V[f_k] = \frac{\sum_{i=0}^{N_{FFT}-1} w_i x_i e^{-j2\pi ik/N_{FFT}}}{\sum_{i=0}^{N_{FFT}-1} w_i}, \qquad k = 0, 1, \ldots, N_{FFT}-1$$

where $V[f_k]$ is the amplitude at frequency f_k, x_i is the sequence of measured values and w_i is the optional FFT windowing function. For $w_i = 1$, $i = 0, 1, ..., N$ -1 (no windowing), the voltage spectrum equation is the normal discrete Fourier transform.

If frequency folding is being performed, the folding is accomplished by adding the complex conjugate of the negative frequency term to the corresponding positive frequency bin. If the negative frequency terms do not correspond exactly to a positive frequency term, the conjugate term is added proportionately to the two positive frequency bins closest to the desired frequency using:

$$V'[f_k] = V[f_k] + V^*[-f_l] \cdot (-f_l - f_k)/(f_{k+1} - f_k)$$

$$V'[f_{k+1}] = V[f_{k+1}] + V^*[-f_l] \cdot (f_{k+1} + f_l)/(f_{k+1} - f_k)$$

where f_l is the negative frequency and f_k, f_{k+1} are the two positive frequency bins closest to $-f_l$, and $f_k < -f_l < f_{k+1}$.

When averaging is being performed, the average voltage spectrum is computed by averaging the magnitudes of several DFTs:

$$V[f_k]_{Avg} = \frac{1}{N_{Avg}} \cdot \sum_{N_{Avg}} |V[f_k]|$$

where each subsequent DFT is advanced by Overlap Ratio $\cdot$ N_{FFT} samples from the previous DFT.

The **Spectrum Type** setting determines the final values displayed:

Voltage Spectrum: $V[f_k]$ or $V[f_k]_{Avg}$ is displayed.

Threshold Phase: This applies only when the **Angle** option is selected, otherwise the output is identical to "Voltage Spectrum". When $|V[f_k]|$ is very close to zero, the phase at f_k is set to 0.

Normalized Spectrum: $V[f_k]$ or $V[f_k]_{Avg}$ is normalized prior to display. If the **Angle** option is selected, this behaviors similar to "Threshold Phase". If the other options are selected, the normalization depends on whether **dB** is selected. If **dB** is selected the displayed values are offset so the largest spectrum value has magnitude 0 dB. If **dB** is not selected, the spectrum values are linearly scaled so the largest displayed value is displayed as 1 and the smallest spectrum value is displayed as 0.

Unwrap Phase: This applies only when the **Angle** option is selected, otherwise the output is identical to "Voltage Spectrum". This is similar to the "Threshold Phase" option except the angle values are 'unwrapped' relative to the phase at the center frequency (DC for real signals). When unwrapped, adjacent phase values are adjusted by $\pm 2\pi \cdot n$ to keep the difference between the phase values less than or equal to π. Note that when working with tones, this setting may appear to have no effect. This occurs if there are empty frequency bins separating the tones since those bins will have a phase value of 0.

VSD RMS (Vrms/sqrt(Hz)), VSD Peak (Vpeak/sqrt(Hz)): These options display either rms voltage or peak to peak voltage in each FFT bin divided by the square root of the frequency span of the bin.

Spectrum Analyzer Voltage Spectrum: Windowing similar to a spectrum analyzer is applied. The value read on the y-axis corresponds to the frequency content near the frequency bin even when spectral spreading occurs.

The "Threshold Phase" option is useful for reducing phase noise due to round-off errors when the FFT is performed. Because of these round-off errors, frequency bins whose values should be zero often contain very small values rather than zero. If phase is being displayed, computing phase from these values results in arbitrary phase values. The "Threshold Phase" option operates normalizing the voltage spectrum by dividing each value by the norm of the largest magnitude value. A phase of

0 is then displayed for any frequency bin whose magnitude is less than 10^{-10}.

Complementary Cumulative Distribution Function: CCDF

Summary

CCDF computes the power complementary cumulative distribution (CCDF) function from a time domain signal. The CCDF curve shows the amount of time a signal spends above the average power level of the measured signal, or equivalently, the probability that the signal power will be above the average power level.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Number of Bins	Integer value	>=2
Lower Threshold	Real value	<Upper Threshold
Upper Threshold	Real value	>Lower Threshold
Data Window in Samples	Integer value	>0
Sample Accumulation	List of options	N/A
Threshold Limit Handling	List of options	N/A
Y Axis Format	List of options	N/A
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns real values.

Graph Type

This measurement can be displayed on a rectangular graph or tabular grid.

Computational Details

The CCDF curve is generated by first computing the probability density function (PDF) using a histogram of the measured signal. The number of samples histogramed is determined by the **Data Window in Samples** and **Sample Accumulation** parameters. If Sample Accumulation is:

- Only Data Window Samples: The number of samples is **Data Window in Samples**.
- Cumulative: All the samples up to the current measured time are used (the current measured time is determined by the slider bar).

The samples are sorted by instantaneous power into **Number of Bins** bins. The thresholds of the bins are calculated from:

$$P_{bin,\,min} = LT + bin \cdot (UT - LT)/N \qquad bin = 0, 1, \ldots, N-1$$

where $P_{bin,min}$ is the smallest power value allowed in the bin, *bin* is the bin number, LT is the **Lower Threshold** parameter, UT is the **Upper Threshold** parameter, and N is the **Number of Bins** parameter.

Samples that fall below the **Lower Threshold** value or are greater than or equal to the **Upper Threshold** value are included in bin 0 and bin N-1, respectively, if **Threshold Limit Handling** is set to “Include all samples”. If **Threshold Limit Handling** is set to “Exclude samples beyond threshold limits” these samples are not counted.

The PDF is then calculated for each bin by dividing the number of samples in the bin by the total number of samples counted. The cumulative distribution function (CDF), which is the integral of the PDF, is then calculated by:

$$CDF[bin] = \sum_{i=0}^{bin} PDF[i]$$

The full CCDF is 1-CDF, which estimates the probability of a sample having an instantaneous power value greater than the maximum power in a given bin.

The CCDF measurement only displays the portion of the CCDF for power values greater than the average power of the histogramed samples (any samples discarded because they are outside the threshold range are not included in the average power calculation.) The y-axis values displayed are the probabilities that a sample's instantaneous power will be greater than or equal to the power that is the x-axis value from the average power.

Probability Density Function: PDF

Summary

PDF computes the power probability density function (PDF) a time domain signal.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Number of Bins	Integer value	>=2
Lower Threshold	Real value	<Upper Threshold
Upper Threshold	Real value	>Lower Threshold
Data Window in Samples	Integer value	>0
Sample Accumulation	List of options	N/A
Threshold Limit Handling	List of options	N/A
Y Axis Format	List of options	N/A
Power Display	List of options	N/A
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns real values.

Graph Type

This measurement can be displayed on a histogram graph, rectangular graph, or tabular grid.

Computational Details

The power probability density function (PDF) is computed by generating a a histogram of the measured signal. The number of samples histogramed is determined by the **Data Window in Samples** and **Sample Accumulation** parameters. If **Sample Accumulation** is:

- Only Data Window Samples: The number of samples is **Data Window in Samples**.
- Cumulative: All the samples up to the current measured time are used (the current measured time is determined by the slider bar).

The samples are sorted by instantaneous power into **Number of Bins** bins. The thresholds of the bins are calculated from:

$$P_{bin,\,min} = LT + bin \cdot (UT - LT)/N \qquad bin = 0, 1, \ldots, N-1$$

where $P_{bin,min}$ is the smallest power value allowed in the bin, *bin* is the bin number, LT is the **Lower Threshold** parameter, UT is the **Upper Threshold** parameter, and N is the **Number of Bins** parameter.

Samples that fall below the **Lower Threshold** value or are greater than or equal to the **Upper Threshold** value are included in bin 0 and bin N-1, respectively, if **Threshold Limit Handling** is set to “Include all samples”. If **Threshold Limit Handling** is set to “Exclude samples beyond threshold limits” these samples are not counted.

The PDF is then calculated for each bin by dividing the number of samples in the bin by the total number of samples counted.

The x-axis can be set to display either the average power represented by each bin, or the average bin power relative to the average power of the histogramed samples (any samples discarded because they are outside the threshold range are not included in the average power calculation.) This is done through the **Power Display** parameter.

Histogram: HIST

Summary

HIST generates a histogram function from a system simulation signal. It sorts the samples into fixed width bins, producing either the number of samples in each bin or the proportion of samples in each bin to the total number of samples counted.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Number of Bins	Integer value	>=2
Lower Threshold	Real value	<Upper Threshold
Upper Threshold	Real value	>Lower Threshold
Data Window in Samples	Integer value	>0
Sample Accumulation	List of options	N/A
Threshold Limit Handling	List of options	N/A
Y Axis Format	List of options	N/A
X Axis Format	List of options	N/A
*Start Offset	Real value	>= 0
*Signal Delay	List of options	N/A

** indicates a secondary parameter*

NOTE. If the selected system diagram is configured for a swept simulation, the measurement will have additional parameters for specifying the plotting configuration for each swept parameter. These parameters are dynamic and change based upon which data source is selected. See the "*Sweeping Simulations*" section of the "*Simulation Basics*" chapter of the *VSS Modeling Guide* for details.

Result

This measurement returns real or complex values.

Graph Type

This measurement can be displayed on a histogram graph, rectangular graph, or tabular grid.

Computational Details

This measurement calculates the histogram by first generating a set of **Number of Bins** bins according to:

$$T_{bin,\,min} = LT + bin \cdot (UT - LT)/N \qquad bin = 0, 1, \ldots, N-1$$

where $T_{bin,min}$ is the smallest value allowed in the bin, *bin* is the bin number, LT is the **Lower Threshold** parameter, UT is the **Upper Threshold** parameter, and N is the **Number of Bins** parameter.

Samples that fall below the **Lower Threshold** value or are greater than or equal to the **Upper Threshold** value are included in bin 0 and bin N-1, respectively, if **Threshold Limit Handling** is set to “Include all samples”. If **Threshold Limit Handling** is set to “Exclude samples beyond threshold limits” these samples are not counted.

The number of samples histogrammed is determined by the **Data Window in Samples** and **Sample Accumulation** parameters. If **Sample Accumulation** is:

- Only Data Window Samples: The number of samples is **Data Window in Samples**.
- Cumulative: All the samples up to the current measured time are used (the current measured time is determined by the slider bar).

The **X Axis Format** parameter controls the value displayed along the x-axis corresponding to the individual bins.

- Center of bin: Displays the average of the lower and upper thresholds of the bin.
- Lower edge of bin: Displays the lower threshold of the bin.
- Upper edge of bin: Displays the upper threshold of the bin.
- Bin number, from 1: Displays the bin number, the first bin is numbered ‘1’.
- Bin number, from 0: Displays the bin number, the first bin is numbered ‘0’.

Static Signal Delay Properties: DLY_PROP

Summary

DLY_PROP displays the static signal delay property of a signal. The static properties of a signal are computed prior to the generation of data samples. The static signal delay property is an estimate of the signal delay imparted by various blocks on the signal, as determined by the individual blocks.

For example, FIR filter blocks introduce a signal delay equal to the position of the peak of the magnitude impulse response of the filter. Delay blocks introduce a signal delay corresponding directly to the block's delay. The static signal delay property is then the sum of all the static signal delays introduced by blocks along the signal path.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Output Type	List of options	N/A

Result

This measurement returns unitless real values.

Graph Type

This measurement can be displayed on a tabular grid.

Static Power-Noise Ratio Properties: EsN0_PROP

Summary

EsN0_PROP displays one of several power-noise ratios of a signal based on the static properties of a signal. The static properties of a signal are computed prior to the generation of data samples.

The power-noise ratios are computed from the static signal power property and the static noise property. These properties are viewable using the *PWR_PROP* and *NOISE_PROP* measurements.

NOTE. The static power-noise ratios are intended to be used primarily when performing BER measurements. They are used by the BER meter and measurements to determine values for the x-axis if none are explicitly specified.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Output Type	List of options	N/A

Result

This measurement returns unitless real values.

Graph Type

This measurement can be displayed on a tabular grid.

Frequency Properties: FRQ_PROP

Summary

FRQ_PROP displays the frequency properties of a signal, such as sampling frequency, data rate, and center frequency.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Output Type	List of options	N/A

Result

This measurement returns real values in frequency units. The frequency value displayed is determined by the **Output Type** setting.

Graph Type

This measurement can be displayed on a tabular grid.

Static Noise Properties: NOISE_PROP

Summary

NOISE_PROP displays the static noise property of a signal. The static properties of a signal are computed prior to the generation of data samples. The static noise property is an estimate of the total noise imparted by various blocks on the signal, as determined by the individual blocks.

For example, the Additive White Gaussian Noise block, AWGN, adds its noise level to the signal's noise property. RF blocks that support noise figure, such as AMP_B or NL_S, also add the noise level of their generated noise to the signal's noise property. RF blocks that apply gain or attenuation to the signal also adjust the noise property, since the noise is also amplified/attenuated along with the main signal.

The static noise property is normally used by the Bit Error Rate and Symbol Error Rate blocks (BER, BER_EXT, SER, and SER_EXT) when automatically computing the Es/N0 or Eb/N0 ratios.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Output Type	List of options	N/A

Result

This measurement returns real values in power units. The **Output Type** setting selects the representation of the noise value. These settings are compatible with the AWGN block's PWRTYP parameter. See the AWGN documentation for details on the individual settings.

Graph Type

This measurement can be displayed on a tabular grid.

Static Phase Rotation Properties: PHS_PROP

Summary

PHS_PROP displays the static phase rotation property of a signal. The static properties of a signal are computed prior to the generation of data samples. The static phase rotation property is an estimate of the cumulative phase rotation imparted by various blocks on the signal, as determined by the individual blocks.

In general, when estimating phase rotation a block uses the static signal power property to synthesize a sample, apply the block's transfer function to the sample, then measure the change in phase of the generated sample. The change in phase is then added to the phase rotation property of the input signal to obtain the new phase rotation property.

The static phase rotation property is normally used by the receiver blocks to de-rotate the signal prior to demodulation and detection. The EVM measurement also uses it to perform phase compensation.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A

Result

This measurement returns real values with angle units.

Graph Type

This measurement can be displayed on a tabular grid.

Static Signal Power Properties: PWR_PROP

Summary

PWR_PROP displays the static signal power property of a signal. The static properties of a signal are computed prior to the generation of data samples. The static signal power property is an estimate of the signal power generated by a power based source block such as a transmitter or a tone source. Other blocks typically adjust the power property if they apply gain or attenuation to the signal. In general, only the RF and signal processing blocks that affect the signal power level adjust the property.

Blocks whose gain is affected by the input signal's power level, such as the nonlinear amplifiers and mixers, typically use the static power property to synthesize a single sample, apply the block's transfer function, and adjust the power property according to the power of the output sample.

The static signal power property is normally used by the receiver blocks to determine the signal gain. The EVM measurement also uses it to perform magnitude compensation. The Bit Error Rate and Symbol Error Rate blocks also use the static signal power property when automatically determining Es/N0 or Eb/N0.

Parameters

Name	Type	Range
Block Diagram	System Diagram	N/A
Test Point	System Meter	N/A
Output Type	List of options	N/A

Result

This measurement returns real values in units of power. The Output Type setting selects the representation of the power level. The normalized Es and Eb settings correspond to the Normalized Es and Eb settings found in most of the transmitter blocks. See the QAM Transmitter block (QAM_TX) documentation for details.

Graph Type

This measurement can be displayed on a tabular grid.

INDEX

T

U

V

W

Z