Microwave Office®/Analog Office® 2007

Measurement Reference

AWR

Microwave Office/Analog Office 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

The AWR® Design Environment™ suite incorporating Microwave Office® and Analog Office® software is a powerful fully-integrated design and analysis tool for RF, microwave, millimeterwave, analog, and RFIC design.

Microwave Office and Analog Office allow you to design complex circuit designs composed of linear, nonlinear, and EM structures, and generate layout representations of these designs. They allow you to perform fast and accurate analysis of your designs using linear, nonlinear harmonic balance, nonlinear Volterra-series, electromagnetic (EM), and HSPICE simulation engines, and feature real-time tuning and optimizing capabilities.

ABOUT THIS BOOK

This book provides complete reference information on all of the measurements (i.e., computed data such as gain, noise, power, or voltage) that you can choose as output for your linear, nonlinear, EM, and HSPICE simulations.

The measurements in this guide are organized alphabetically in categories such as “Electromagnetic”, “Linear Gain”, and “Nonlinear Power” (also organized alphabetically). This organization reflects how they are displayed in Microwave Office/Analog Office in the **Meas.Type** and **Measurement** fields of the Add Measurement dialog box.

For each measurement in this guide, the following attributes are described:

Attribute	Description
Measurement Type	General category of measurement, such as “Electromagnetic”, “Linear Gain”, or “Nonlinear Power”. For an overview of the measurement categories, see the *Microwave Office/Analog Office User Guide*.
Description	Describes the measurement and provides information on what this measurement is typically used for.

Attribute	Description
Measurement Parameters	The user-modifiable input parameters, such as data source name, port index, and power sweep index, 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 rectangular, polar graph, or Smith chart, on which this measurement can be charted. For an overview of the graph types, see the *Microwave Office/Analog Office User Guide*.
Options	If applicable, indicates the post-processing available for this measurement.

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

Additional Documentation

Microwave Office and Analog Office include the following additional documentation:

- *What's New in MWO/AO 2007?* presents the new features, elements, and measurements for this release.
- *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.

- *Microwave Office/Analog Office User Guide* describes how to use the Microwave Office/Analog Office windows, menu choices, and dialog boxes to perform linear, nonlinear, and EM design, layout, and simulation, and discusses related concepts.
- *Microwave Office/Analog Office Element Catalog Volumes 1, 2,* and *3* provides complete reference information on the electrical element model database that you use to build schematics.
- *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 and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips.
- *Known Issues* lists the known issues for this release.

This guide uses the following typographical conventions.

Item	Convention
Anything that you select (or click on) in the Microwave Office/Analog Office design environment, like 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 within 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	Shown in italics: See the *DEFAULTS.LPF* file.
Any field within a file	Shown in an alternate bold type: Define this parameter in the **$DEFAULT_VALUES** field.

GETTING ON-LINE HELP

Microwave Office/Analog Office online Help provides information as you need it on the windows, menu choices, and dialog boxes that compose the design environment, as well as on the concepts involved.

To access context-sensitive help for each measurement:

- After creating a graph, select **Project > Add Measurement** from the pull-down menu. Select the measurement type and measurement of interest from the **Meas. Type** and **Measurement** scroll boxes, and click the **Meas. Help** button.

Annotate DC Values to V_PROBE, I_METER, and V_Meter: DCA_M

Summary

DCA_M measures the DC voltages at V_PROBE and V_METER and DC current through I_METER in the circuit.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Display type	List of options	On Device or Tool Tip

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage or current.

Annotate DC Input Current for All Elements: DCIA

Summary

DCIA measures the DC input current for all of the elements in the circuit.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Zero threshold (A)	Real	N/A
Search pattern[a]	String	N/A

a Search Pattern uses the Name and ID of the elements in the form of “Name.ID” while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units.

Annotate Current Density: DC_IDENSA

Summary

DC_IDENSA measures the current density in a circuit element or a model specified by an element **Width parameter**. It measures current and uses the **Width parameter** to calculate the density. If the measured density is larger than the **Maximum Allowed** density, the value displays in red with a simulator error message.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Maximum Allowed (A/m)	Real	N/A
Width parameter	String	N/A
Search pattern[a]	String	N/A

a Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in ampere per meter.

Annotate DC Terminal Current on the Element: DC_IE

Summary

DC_IE measures the DC input current the specified elements in the circuit. The current is displayed for each model, not at each node as is done in the DCIA measurement.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Zero threshold (A)	Real	N/A
Search pattern[a]	String	N/A
Display type	List of options	On Device or Tool Tip
Annotate for	List of options	Nonlinear devices or All elements

a Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units. The **Display type** parameter will determine if the power is displayed on the schematic or when the cursor is over a model in the schematic. The **Annotate for** parameter will select every element or just nonlinear devices.

Annotate DC Power for All Elements: DC_PWRA

Summary

DC_PWRA measures DC power across an element or model in a circuit. The **Display type** parameter will determine if the power is displayed on the schematic or when the cursor is over a model in the schematic.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Zero threshold (W)	Real	N/A
Search pattern[a]	String	N/A
Display type	List of options	On Device or Tool Tip

a Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in watts.

Annotate DC Voltage for 2-port Elements: DCVA_E

Summary

DCVA_E measures the DC voltage for all of the 2-port models in the circuit.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Display type	List of options	Value or Meter
Minimum Voltage	Real	N/A
Maximum Voltage	Real	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units either in numerical form or displays in volt meter.

Options

The **Display type** setting controls how the annotation looks on the schematic. For the **Value** setting, the absolute value is displayed on the schematic and the **Minimum Voltage** and **Maximum Voltage** have no effect. For the **Meter** setting, the value on a meter with the **Minimum Voltage** and **Maximum Voltage** as the meter range**.**

Annotate DC Voltage for All Nodes: DCVA_N

Summary

DCVA_N measures the DC voltage at every node in the circuit.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Search pattern[a]	String	N/A
Display type	List of options	Value or Meter
Minimum Voltage	Real	N/A
Maximum Voltage	Real	N/A

a Search Pattern uses the Name and ID of the elements in the form of “Name.ID” while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units either in numerical form or displays in volt meter.

Options

The **Display type** setting controls how the annotation looks on the schematic. For the **Value** setting, the absolute value is displayed on the schematic and the **Minimum Voltage** and **Maximum Voltage** have no effect. For the **Meter** setting, the value on a meter with the **Minimum Voltage** and **Maximum Voltage** as the meter range.

Annotate DC Terminal Voltage on the Element: DC_VE

Summary

DC_VE measures the DC voltage on all the terminals of elements and subcircuits.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	0-1000
Zero threshold (W)	Real	N/A
Search Pattern[a]	String	N/A
Display Type	List of Options	On Device or Tool Tip
Annotate for	List of Options	Nonlinear Devices or All Elements

a Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The **Display type** parameter will determine if the power is displayed on the schematic or when the cursor is over a model in the schematic. The **Annotate for** parameter will select every element or just nonlinear devices.

Current in INets: INET_I

Summary

INET_I back-annotates the current flowing in layout nets that are drawn using Inet. **View Number** indicates the schematic layout view number, for example if it is set to 2 and two layout views are opened, the measurement is applied to the second layout view. **Symbol size** determines the display text size.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
View Number	Integer	N/A
Symbol size (m)	Real	N/A
Specify max density (Amps/m^2)	Real	N/A
Selected nets only	Selection	N/A
Display segment current	Selection	N/A
Display via current	Selection	N/A
Automatically Update	Selection	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is displayed in the schematic Layout View. If **Selected nets only** is selected the nets need to be selected to display the result in the Layout View after simulation.

Current Density in INets: INET_J

Summary

INET_J back-annotates the current density in layout nets that are drawn using Inet. **View Number** indicates the schematic Layout View number, for example if V**iew Number** is set to 2 and two layout views are opened, the measurement is applied to the second layout view.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
View Number	Integer	N/A
Symbol size (m)	Real	N/A
Selected nets only	Selection	N/A
Display segment current	Selection	N/A
Display via current	Selection	N/A
Automatically update	Selection	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is displayed on the schematic Layout View. If **Selected nets only** is selected the nets need to be selected to display the result in the Layout View after simulation.

INet Parasitic Estimate: INET_RC

Summary

INET_RC displays the Parasitic Resistance and Capacitance values of the layout nets that are drawn using Inet. **View Number** indicates the schematic Layout View number, for example, if the View Number is set to 2 and two Layout Views are opened, the measurement is applied to the second Layout View. **Symbol size** determines the size of the parasitic symbols.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
View Number	Integer	N/A
Symbol size (m)	Real	N/A
Selected nets only	Selection	N/A
Total net capacitance	Real	N/A
Show net capacitance	Selection	N/A
Show Route Resistance	Selection	N/A
Automatically Update	Selection	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is displayed on the schematic Layout View. If **Selected nets only** is selected the nets must be selected to display the result in the Layout View after simulation.

Noise Contribution for Elements: NoiseConA

Summary

NoiseConA calculates the amount of noise each element contributes to the noise measured in a circuit using a noise meter (V_NSMTR) specified in **Meter for Noise Meas.** The **Noise Type** is either **Total Noise** or **All Noise**. This measurement is only applied to HSPICE AC and APLAC AC analysis.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Meter for Noise Measure-ment	String	N/A
Noise type	Selection	All Noise/Total Noise
Display type	Selection	On Device/Tool Tip

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is annotated in the schematic either as a tool tip or on device.

Annotate Operating Points of Nonlinear DC Simulator for Elements: OpPnt_DC_E

Summary

OpPnt_DC_E displays all the operating points, voltages, and currents for elements.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Display op points	Real	N/A
Display currents	Real	N/A
Display voltages	Real	N/A
Display secondary op points	Real	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value and is displayed as a tool tip only.

Annotate Single Operating Point Value for Elements: OpPnt1_DC_E

Summary

OpPnt1_DC_E displays a single operating point for elements. Use the component browser to see the list of available operating points.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Select Operating Point	List of operating points	N/A
Display type	List of options	On Device or Tool Tip

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value and is displayed on the device in the schematic or as a tool tip.

Annotate Total Power for All Elements: TOT_PWRA

Summary

TOT_PWRA measures RMS power across an element or model in a circuit.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Zero threshold (W)	Real	N/A
Search pattern[a]	String	N/A

a Search Pattern uses the Name and ID of the elements in the form of “Name.ID” while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in dBm.

Annotate Vtime Measurement for 2 Port Elements: VTimeA_E

Summary

VTimeA_E displays a time voltage waveform at each node of a two port element in the schematic.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Minimum Voltage	Real	N/A
Maximum Voltage	Real	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement displays a voltage waveform.

Annotate Voltage at Time Point for All Nodes: VTPA_N

Summary

VTPA_N measures instantaneous voltage at a given time for each node.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Search pattern[a]	String	N/A
Time point	Real	N/A
Minimum Voltage	Real	N/A
Maximum Voltage	Real	N/A

a Search Pattern uses the Name and ID of the elements in the form of “Name.ID” while searching. For example, to search for all resistors you would type **RES*** as the search pattern.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in volts displayed in numerical form or in a meter.

Operating Point ERC Rule: ERC_OP

Summary

ERC_OP verifies whether or not the operating point of a device/devices selected in **Select Operating Point** is within the specified range of the **Lower Limit** and **Upper Limit**.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Search Pattern[1]	String	N/A
Select Operating Point	String	N/A
Upper Limit	Real	N/A
Lower Limit	Real	N/A

1 Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for GBJT with ID=Q3, you would type GBJT.Q3 as the search pattern. "" denotes the wildcard and searches for all the iNets in layout.*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The ERC rule violation is displayed in the ERC violation window. This window displays only when there is an ERC violation.

Draw Parasitics in 3D: EXT_CKT3D

Summary

EXT_CKT3D displays the parasitic values of the iNet in the 3D view of the extracted layout. **View Number** indicates the 3D Layout View number. For example, if **View Number** is set to 2 and two 3D Layout Views are opened, the measurement is applied to the second 3D Layout View. **Symbol size** determines the size of the parasitics symbols.

Parameters

Name	Type	Range
EM Simulation Document	EM Document	N/A
View Number	Integer	N/A
Symbol size (m)	Real	N/A
Display Values	Real	N/A
Show resistors	Selection	N/A
Show capacitors	Selection	N/A
Show inductors	Selection	N/A
Show series	Selection	N/A
Show shunt	Selection	N/A
Show Coupling	Selection	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is displayed on the schematic Layout View. If **Selected nets only** is selected, the nets need to be selected to display the result in the Layout View after simulation.

Max Current ERC Rule: ICHECK

Summary

ICHECK verifies whether or not the current in an iNet/iNets is above the specified **Max Current (Amps)**.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Search pattern[1]	String	N/A
Max Current (Amps)	Real	N/A

1 Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for Inet with ID EN3 you would type ELENET2.EN3 as the search pattern. "" denotes the wildcard and searches for all the Inets in layout.*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The ERC rule violation is displayed in an ERC violation window. This window displays only when there is an ERC violation.

Current Density ERC Rule: JCHECK

Summary

JCHECK verifies whether or not the current density in an iNet/iNets is above the specified **Max Current Density (Amps/m2)**.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Search pattern[1]	String	N/A
Max Current Density (Amps/m^2)	Real	N/A

*1 Search Pattern uses the Name and ID of the elements in the form of "Name.ID" while searching. For example, to search for Inet with ID EN3 you would type ELENET2.EN3 as the search pattern. * denotes the wildcard and searches for all the Inets in layout.*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The ERC rule violation is displayed in an ERC violation window. This window displays only when there is ERC violation.

Total Noise Contribution ERC: NoiseCheck

Summary

NoiseCheck verifies whether or not the total noise contribution from all the elements measured with the noise voltage meter (V_NSMTR) specified in **Meter for Noise Meas.** is above the specified **Maximum Noise Voltage (V2/Hz)**.

Parameters

Name	Type	Range
Top Level Schematic	Subcircuit	N/A
Meter for Noise Measurement	String	N/A
Maximum Noise v^2/hz	Real	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The ERC rule violation is displayed in an ERC violation window. This window displays only when there is an ERC violation.

EM Document Current: EM_CURRENT

Summary

EM_CURRENT displays a representation of the electric current occurring on the specified 3D view of a specified EM document. The current is overlaid with the structure drawn in wireframe mode. This annotation allows you to specify the frequency, phase, vector components and color scaling associated with the magnitude of the current.

Parameters

Name	Type	Range
Em Data Source	EM document	N/A
View Number	N/A	N/A
Include X-Component	N/A	N/A
Include Y-Component	N/A	N/A
Include Z-Component	N/A	N/A
Phase Index	Real	N/A
Show Current Directions	N/A	N/A
Show Color Bar	N/A	N/A
Use Data Smoothing	N/A	N/A
Log Range (dB)	N/A	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns amperes per meter (A/m).

EM Document Electric Field: EM_E_FIELD

Summary

EM_E_FIELD allows display of a representation of the electric field occurring at a specific layer on the specified 3D view of a specified EM document. The electric field is overlaid with the structure drawn in wireframe mode. This annotation allows you to specify the frequency, phase, layer, vector components and color scaling associated with the magnitude of the electric field.

Parameters

Name	Type	Range
EM Data Source	EM document	N/A
View Number	N/A	N/A
Include X-Component	N/A	N/A
Include Y-Component	N/A	N/A
Layer Number	N/A	N/A
Phase Index	N/A	N/A
Show Field Directions	N/A	N/A
Show Color Bar	N/A	N/A
Use Data Smoothing	N/A	N/A
Log Range (dB)	Real	1 -100

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement displays volts per meter (V/m).

EM Document Mesh: EM_MESH

Summary

EM_MESH allows display of the discretization (mesh) on the specified 3D view of a specified EM document. The current is overlaid with the structure drawn in wire-frame mode. This annotation allows you to control if the mesh is synchronized with changes in the input data structures.

Parameters

Name	Type	Range
EM Simulation Document	EM document	N/A
View Number	N/A	N/A
Reference Edges Only	N/A	N/A
Synchronize Mesh	N/A	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

EM Document Mesh 2D: EM_MESH_2D

Summary

EM_MESH_2D allows display of the discretization (mesh) on the specified 2D view of a specified EM document. The current is overlaid with the structure drawn in wireframe mode. This annotation allows you to control if the mesh is synchronized with changes in the input data structures.

Parameters

Name	Type	Range
EM Simulation Document	EM document	N/A
View Number	N/A	N/A
Reference Edges Only	N/A	N/A
Synchronize Mesh	N/A	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Plot Column of X-Y Data: PlotCol

Summary

PlotCol plots a column of real X-Y data specified in a tabular text file. You specify the column representing the x-axis and the column representing the y-axis.

Parameters

Name	Type	Range
Data File Name	Subcircuit	Text file contains two or more columns
Column for X Axis	Integer	1 to 1000
Column for Y Axis	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Data measurements are based off of data files and so swept parameters are not defined.

Result

PlotCol returns a real value. You can also display the value in dB by selecting the **DB** check box under **Result Type**. You specify a unitless x-axis for this measurement.

Computational Details

For each value in the specified x-axis column, the corresponding value in the specified y-axis column is plotted. The number of points in each column must be equal.

The text file must contain at least two columns. 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:

! Example of a tabular, text data file for use with PlotCol

1.055 0.4570.689

2.110 1.2561.941

! End of file

Plot Row of X-Y Data: PlotRow

Summary

PlotRow plots a row of real X-Y data specified in a tabular text file. You specify the row representing the x-axis and the row representing the y-axis.

Parameters

Name	Type	Range
Data File Name	Subcircuit	Text file contains two or more columns
Row for X Axis	Integer	1 to 1000
Row for Y Axis	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Data measurements are based off of data files and so swept parameters are not defined.

Result

PlotRow returns a real value. You can also display the value in dB by selecting the **DB** check box under **Results Type**. You specify the unitless x-axis for this measurement.

Computational Details

For each value in the specified x-axis row, the corresponding value in the specified y-axis row is plotted. The number of points in each row must be equal.

The text file must contain at least two rows. Multiple values on the same row must be tab-separated. An exclamation point at the beginning of a line indicates a comment line. An example of a data file is shown below:

! Example of a tabular, text data file for use with PlotRow

1.055 0.4570.689

2.110 1.2561.941

! End of file

EM Mode Resonance (Composite): AllModes

Summary

The AllModes measurement creates a sum of all TM_Mode[m,n] and TE_Mode[m,n] measurements where all modes used for the sum have m and n less than or equal to the Max Mode parameter that was set during the measurement's creation. The AllModes measurement can be used to find resonances in the enclosure without having to look at every possible mode that may be resonant.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Max Mode	Integer	0 to 99

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.The x-axis for this measurement is always in frequency units.

Port Relative Dielectric Constant: Er_Port

Summary

The Er_Port measurement is used to compute the relative dielectric constant of the section of line leading up to a port. When an Er_Port measurement is created, the port number must be specified. Deembedding must be enabled when measuring the port propagation constant. If deembedding is not enabled, then the relative dielectric constant measurement will issue an error message. The Er_Port measurement only works for one port on a side. If there is more than one port on a side then the relative dielectric constant measurement will issue an error message indicating this condition.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Port Number	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

X-Directed E-Field (Volt/Meter): Ex_EM

Summary

The Ex_EM measurement calculates the x-directed electric field density (Volt/meter) along a cross sectional cut of an EM structure at a specified layer and frequency. The vertical cross section can be specified as a constant x or y dimension, specified in cell units. This measurement quantizes the x directed E-field seen in an EM animation along the aforementioned cut. Note that the measurement values will be effected by all EM port properties, including excitation amplitude, phase and termination values.

Parameters

Name	Type	Range
EM Data Source	EM Simulation	1 to 1000 ports
Frequency Index	Integer	1 to # EM Freqs
Layer Number	Integer	1 to # of EM Layers
Cross Sectional Cut Direction	List of Options	Fixed X or Y Coordinate
Dist from Origin to Cut	Real	0 to # EM cells

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 box. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in length units.

Computational Details

The origin for the EM simulator is located in the upper left hand corner when viewed in two dimensions. The positive x direction is to the right of the origin and the positive y direction is below the origin.

Options

The scroll bar available at the bottom of the graph allows the phase of the excitation to be changed from zero (extreme left) to 360 degrees (extreme right), which is the range of phase values seen in the EM animation.

Y-Directed E-Field (Volt/Meter): Ey_EM

Summary

The Ey_EM measurement calculates the y-directed electric field density (volt/meter) along a cross sectional cut of an EM structure at a specified layer and frequency. The vertical cross section can be specified as a constant x or y dimension, specified in cell units. This measurement quantizes the y directed E-field seen in an EM animation along the aforementioned cut. Note that the measurement values will be effected by all EM port properties, including excitation amplitude, phase and termination values.

Parameters

Name	Type	Range
EM Data Source	EM Simulation	1 to 1000 ports
Frequency Index	Integer	1 to # EM Freqs
Layer Number	Integer	1 to # of EM Layers
Cross Sectional Cut Direction	List of options	Fixed X or Y Coordinate
Dist from Origin to Cut	Real	0 to # EM cells

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 box. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in length units.

Computational Details

The origin for the EM simulator is located in the upper left hand corner when viewed in two dimensions. The positive x direction is to the right of the origin and the positive y direction is below the origin.

Options

The scroll bar available at the bottom of the graph allows the phase of the excitation to be changed from zero (extreme left) to 360 degrees (extreme right), which is the range of phase values seen in the EM animation.

Fast Frequency Sweep Error: FFS_Error

Summary

The FFS_Error measurement is used to provide an estimate of the accuracy of the FFS solver over the extrapolated frequency band. It is important to note that the FFS error estimate is only an estimate of the error and not the true error. EMSight usually overestimates the error to be conservative, although occasionally the error will be somewhat higher than the estimate (usually at the band edges). EMSight automatically uses the FFS error estimate to truncate the frequency response where the FFS error estimate predicts that the error is above a fixed threshold. The FFS_Error measurement should only be used with the FFS solver. If the FFS_Error measurement is used without using the FFS solver, then the results will be meaningless.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports

Result

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

Graph Type

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

X-Directed Current Density (Amp/Meter): Ix_EM

Summary

The Ix_EM measurement calculates the x-directed current density (Amp/meter) along a cross sectional cut of an EM structure at a specified layer and frequency. The vertical cross section can be specified as a constant x or y dimension, specified in cell units. This measurement quantizes the x directed current seen in an EM animation along the aforementioned cut. Note that the measurement values will be effected by all EM port properties, including excitation amplitude, phase and termination values.

Parameters

Name	Type	Range
EM Data Source	EM Simulation	1 to 1000 ports
Frequency Index	Integer	1 to # EM Freqs
Layer Number	Integer	1 to # of EM Layers
Cross Sectional Cut Direction	List of options	Fixed X or Y Coordinate
Dist from Origin to Cut	Real	0 to # EM cells

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 box. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in length units.

Computational Details

The origin for the EM simulator is located in the upper left hand corner when viewed in two dimensions. The positive x direction is to the right of the origin and the positive y direction is below the origin.

Options

The scroll bar available at the bottom of the graph allows the phase of the excitation to be changed from zero (extreme left) to 360 degrees (extreme right), which is the range of phase values seen in the EM animation.

Y-Directed Current Density (Amp/Meter): Iy_EM

Summary

The Iy_EM measurement calculates the y-directed current density (Amp/meter) along a cross sectional cut of an EM structure at a specified layer and frequency. The vertical cross section can be specified as a constant x or y dimension, specified in cell units. This measurement quantizes the current seen in an EM animation along the aforementioned cut. Note that the measurement values will be effected by all EM port properties, including excitation amplitude, phase and termination values.

Parameters

Name	Type	Range
EM Data Source	EM Simulation	1 to 1000 ports
Frequency Index	Integer	1 to # EM Freqs
Layer Number	Integer	1 to # of EM Layers
Cross Sectional Cut Direction	List of options	Fixed X or Y Coordinate
Dist from Origin to Cut	Real	0 to # EM cells

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 box. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in length units.

Computational Details

The origin for the EM simulator is located in the upper left hand corner when viewed in two dimensions. The positive x direction is to the right of the origin and the positive y direction is below the origin.

Options

The scroll bar available at the bottom of the graph allows the phase of the excitation to be changed from zero (extreme left) to 360 degrees (extreme right) which is the range of phase values seen in the EM animation.

Z-Directed Current Density (Amp/Meter^2): Iz_EM

Summary

The Iz_EM measurement calculates the z-directed current density (Amp/meter^2) along a cross sectional cut of an EM structure at a specified layer and frequency. The vertical cross section can be specified as a constant x or y dimension, specified in cell units. This measurement quantizes the current seen in an EM animation along the aforementioned cut. Note that the measurement values will be effected by all EM port properties, including excitation amplitude, phase and termination values.

Parameters

Name	Type	Range
EM Data Source	EM Simulation	1 to 1000 ports
Frequency Index	Integer	1 to # EM Freqs
Layer Number	Integer	1 to # of EM Layers
Cross Sectional Cut Direction	List of options	Fixed X or Y Coordinate
Dist from Origin to Cut	Real	0 to # EM cells

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 box. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in length units.

Computational Details

The origin for the EM simulator is located in the upper left hand corner when viewed in two dimensions. The positive x direction is to the right of the origin and the positive y direction is below the origin.

Options

The scroll bar available at the bottom of the graph allows the phase of the excitation to be changed from zero (extreme left) to 360 degrees (extreme right) which is the range of phase values seen in the EM animation.

Port Propagation Constant: K_Port

Summary

The K_Port measurement is used to compute the propagation constant for the section of line leading up to a port. When a K_Port measurement is created, the port number must be specified. Deembedding must be enabled when measuring the port propagation constant. If deembedding is not enabled, then the port impedance measurement will issue an error message. The K_Port measurement only works for one port on a side. If there is more than one port on a side then the port impedance measurement will issue an error message indicating this condition.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Port Number	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

TE Mode Resonance: TE_Mode

Summary

The TE_Mode measurement is used to determine if there is a TE mode resonance in the frequency band where the analysis is being performed. The TE_Mode measurement automatically sweeps over a frequency range somewhat larger than the frequency range specified for the EM analysis. The mode index number for the TE mode must be set when the measurement is created (i.e., mode index M and mode index N). The mode measurement is specified as a sum of input impedances Zdown and Zup where the impedances are the corresponding TE mode impedances looking down and up from an arbitrary point in the dielectric stackup. The stackup is viewed as a z-directed set of transmission lines. The enclosure will be resonant when the impedance looking up and the negative of the impedance looking down are equal. Under this condition, the mode measurement will indicate an infinite value (a large finite value is used to represent infinity).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Mode Index M	Integer	0 to 90
Mode Index N	Integer	0 to 90

NOTE. All measurements have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

TM Mode Resonance: TM_Mode

Summary

The TM_Mode measurement is used to determine if there is a TM mode resonance in the frequency band where the analysis is being performed. The TM_Mode measurement automatically sweeps over a frequency range somewhat larger than the frequency range specified for the EM analysis. The mode index number for the TM mode must be set when the measurement is created (i.e., mode index M and mode index N). The mode measurement is specified as a sum of input impedances Zdown and Zup where the impedances are the corresponding TM mode impedances looking down and up from an arbitrary point in the dielectric stackup. The stackup is viewed as a z-directed set of transmission lines. The enclosure is resonant when the impedance looking up and the negative of the impedance looking down are equal. Under this condition, the mode measurement indicates an infinite value (a large finite value is used to represent infinity).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Mode Index M	Integer	1 to 90
Mode Index N	Integer	1 to 90

NOTE. All measurements have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

Port Impedance: Zin_Port

Summary

The Zin_Port measurement is used to compute the impedance of the section of line leading up to a port. When a Zin_Port measurement is created, the port number must be specified. Deembedding must be enabled when measuring the port impedances. If deembedding is not enabled, then the port impedance measurement will issue an error message. The impedance definition used for defining the port impedance is:

$$Z_{port} = \frac{V}{I}$$

where V is the potential from the port conductor to ground and I is the current flowing into the port. The Zin_Port measurement only works for one port on a side. If there is more than one port on a side then the port impedance measurement will issue an error message indicating this condition.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Port Number	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

Conic Axial Ratio (Sweep Phi): Con_AR

Summary

Con_AR calculates axial ratio for a conic cut. The values of Frequency and Theta are fixed while Phi is swept from -180 to 180 degrees or $-\pi$ to π radians.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Theta (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the axial ratio along a conic cut. Although axial ratio is a real value, it is returned as a complex value to remain consistent with the other antenna measurements. The axial ratio is defined as the absolute value of the sum of right-hand circular polarization (RHCP) E-fields and left-hand circular polarization (LHCP) E-fields divided by the difference.

Computational Details

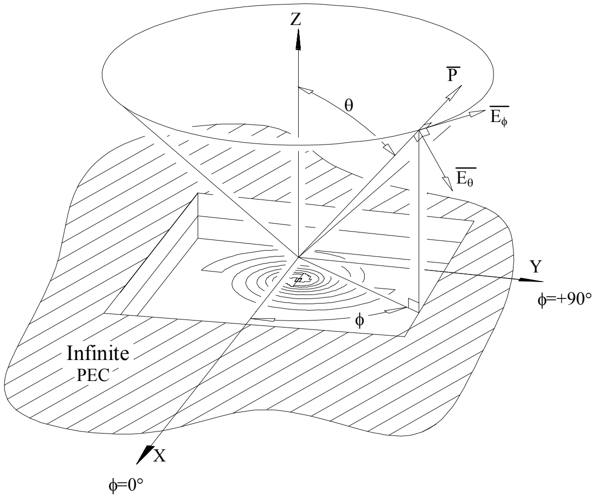
The axial ratio is calculated as:

$$\left|\frac{E_{RHCP} + E_{LHCP}}{E_{RHCP} - E_{LHCP}}\right|$$

E-Phi Pattern (Sweep Phi): Con_EPhi

Summary

Also known as a Conic Cut or Phi Sweep polarized along E_ϕ, this measurement fixes the values of Frequency and Theta while sweeping Phi from -180 to 180 degrees or $-\pi$ to π radians.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Theta (degrees)	Real	0 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result = \left. \frac{E_\phi(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{\theta = const, -180° \le \phi \le 180°}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in this particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

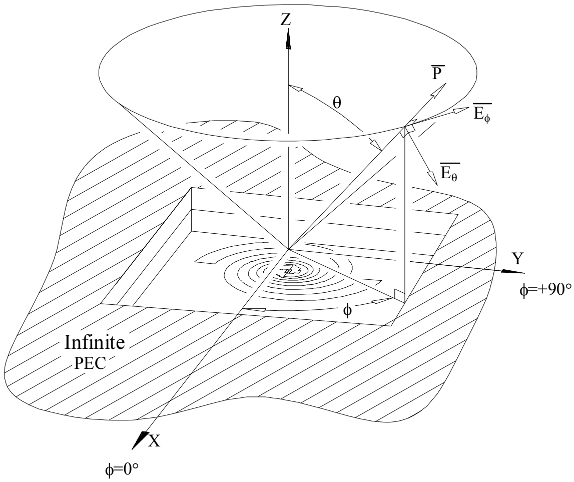
Note

During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

E-Theta Pattern (Sweep Phi): Con_ETheta

Summary

Also known as a Conic Cut or Phi Sweep polarized along E_θ, this measurement fixes the values of Frequency and Theta while sweeping Phi from -180 to 180 degrees or $-\pi$ to π radians.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Theta (degrees)	Real	0 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result = \left. \frac{E_\theta(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{\theta = const, -180° \le \phi \le 180°}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\mathrm{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

Note

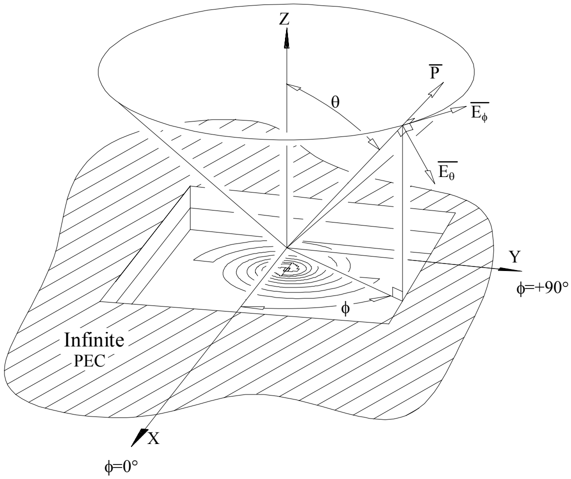
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Left-Hand Circular Polarization (Sweep Phi): Con_LHCP

Summary

Also known as a Conic Cut or Phi Sweep in Left Hand Circular polarization, this measurement fixes the values of Frequency and Theta while sweeping Phi from -180 to 180 degrees or $-\pi$ to π radians. Left Hand Circular Polarization is a linear combination of E_θ and E_ϕ as defined below:

$$LHCP(\theta, \phi) = \frac{E_\theta - jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Theta (degrees)	Real	0 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result = \left. \frac{LHCP(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{\theta = const, -180° \le \phi \le 180°}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

imaginary components can also be displayed on a rectangular graph or a tabular grid.

Note

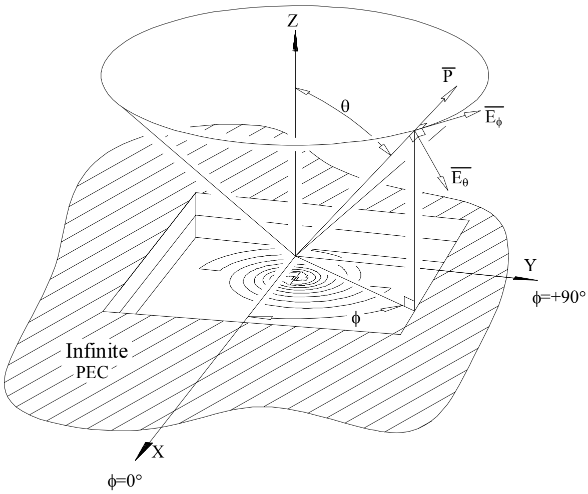
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Right-Hand Circular Polarization (Sweep Phi): Con_RHCP

Summary

Also known as a Conic Cut or Phi Sweep in Right Hand Circular polarization, this measurement fixes the values of Frequency and Theta while sweeping Phi from -180 to 180 degrees or $-\pi$ to π radians. Right Hand Circular Polarization is a linear combination of E_θ and E_ϕ as defined below.

$$RHCP(\theta, \phi) = \frac{E_\theta + jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (theta)	Real	0 to 90
Frequency Sweep Index	Integer e	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result = \left. \frac{RHCP(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{\theta = const, -180° \le \phi \le 180°}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allow one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

Note

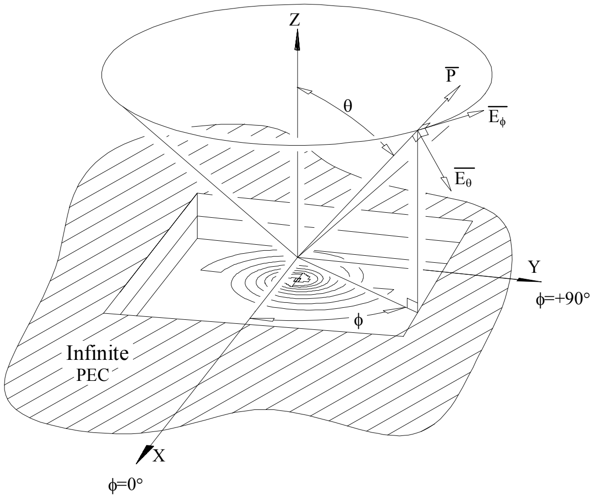
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Total Radiated Power (Sweep Phi): Con_TPwr

Summary

Also known as a Conic Cut or Phi Sweep which captures the total power in all directions, this measurement fixes the values of Frequency and Theta while sweeping Phi from -180 to 180 degrees or $-\pi$ to π radians. The total power is defined as the sum of the power contained in E_θ and E_ϕ :

$$TPwr = \frac{1}{240\pi}(|E_\theta|^2 + |E_\phi|^2)$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Theta (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction for all polarizations. This result is normalized to P_{ave} - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*{}_\phi - E_\phi \cdot H^*{}_\theta) \sin\theta d\theta d\phi$$

$$result = \left.\sqrt{\frac{TPwr(\theta, \phi)}{P_{ave}}}\right|_{-180° \le \phi \le 180°,\ \theta = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction in order to preserve compatibility with other antenna measurements although the measurement is purely real. This insures that $|\text{result}|^2$ is the directivity in that particular direction. The result can be displayed as a real value by specifying the magnitude, or real component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

Note

During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

PPC Axial Ratio (Sweep Theta): PPC_AR

Summary

PPC_AR calculates axial ratio for a principal plane cut. The values of Frequency and Phi are fixed while Theta is swept from -90 to 90 degrees or $-\pi/2$ to $\pi/2$ radians.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the axial ratio along a principal plane cut. Although axial ratio is a real value, it is returned as a complex value to remain consistent with the other antenna measurements. The axial ratio is defined as the absolute value of the sum of right-hand circular polarization (RHCP) E-fields and left-hand circular polarization (LHCP) E-fields divided by the difference.

Computational Details

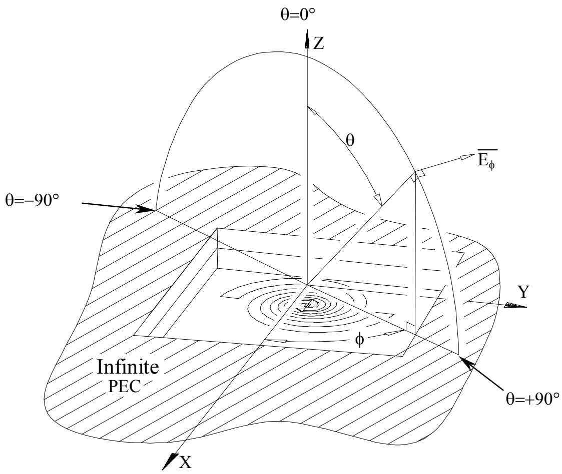
The axial ratio is calculated as:

$$\left|\frac{E_{RHCP}+E_{LHCP}}{E_{RHCP}-E_{LHCP}}\right|$$

E-Phi Pattern (Sweep Theta): PPC_EPhi

Summary

Also known as a Principle Plane Cut or Theta Sweep polarized along E_{ϕ}, this measurement fixes the values of Frequency and Phi while sweeping Theta from -90 to 90 degrees or $-\pi/2$ to $\pi/2$ radians.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result = \left. \frac{E_\varphi(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{-90° \le \theta \le 90°,\ \phi = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

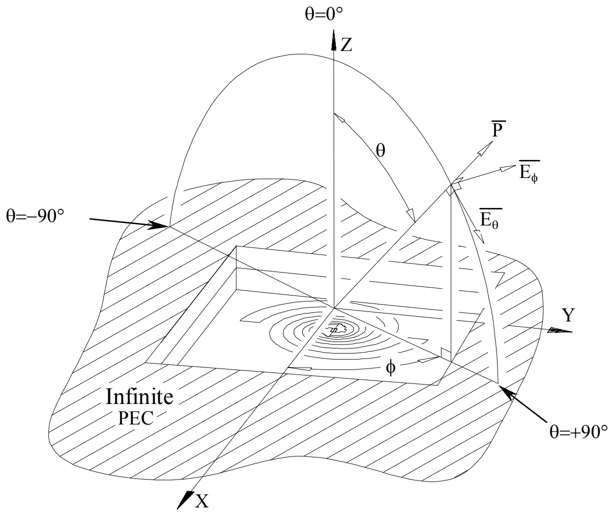
Note

During P_{ave} computation progress bar may be displayed warning user that the lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

E-Theta Pattern (Sweep Theta): PPC_ETheta

Summary

Also known as a Principle Plane Cut or Theta Sweep polarized along E_θ, this measurement fixes the values of Frequency and Phi while sweeping Theta from -90 to 90 degrees or -π /2 to π /2 radians.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result = \left. \frac{E_\theta(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{-90° \le \theta \le 90°, \, \phi = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box 20 log(|result|).The independent axis for this measurement is in angle units.

Note

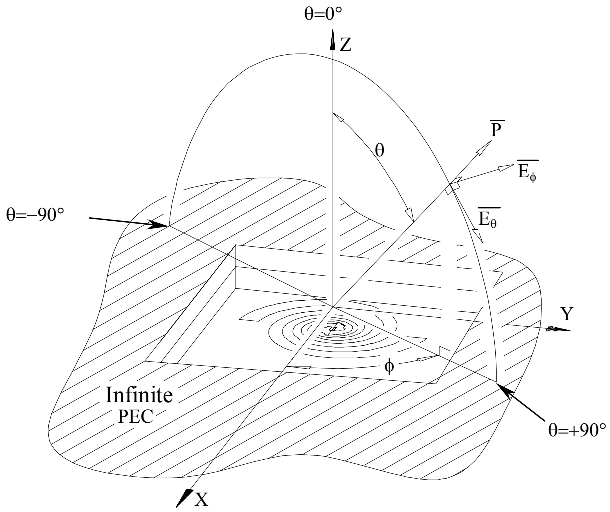
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Left-Hand Circular Polarization (Sweep Theta): PPC_LHCP

Summary

Also known as a Principal Plane Cut or Theta Sweep in Left Hand Circular polarization, this measurement fixes the values of Frequency and Phi while sweeping Theta from -90 to 90 degrees or $-\pi/2$ to $\pi/2$ radians. Left Hand Circular polarization is a linear combination of E_θ and E_ϕ as defined below:

$$LHCP(\theta, \varphi) = \frac{E_\theta - jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result = \left. \frac{LHCP(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{-90° \le \theta \le 90°,\ \phi = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction, such that $|\mathrm{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

NOTE.

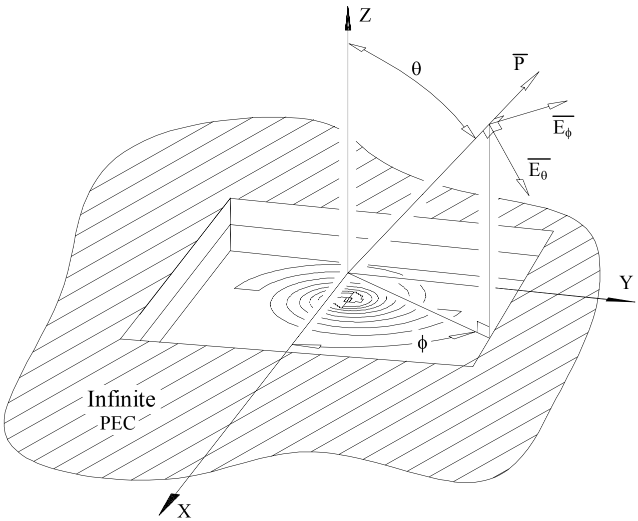
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Right-Hand Circular Polarization (Sweep Theta): PPC_RHCP

Summary

Also known as a Principal Plane Cut or Theta Sweep in Right Hand Circular polarization, this measurement fixes the values of Frequency and Phi while sweeping Theta from -90 to 90 degrees or $-\pi/2$ to $\pi/2$ radians. Right Hand Circular polarization is a linear combination of E_θ and E_ϕ as defined below.

$$RHCP(\theta, \phi) = \frac{E_\theta + jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result = \left. \frac{RHCP(\theta, \phi)}{\sqrt{240\pi P_{ave}}} \right|_{-90° \le \theta \le 90°,\ \phi = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction, such that $|\mathrm{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

Note

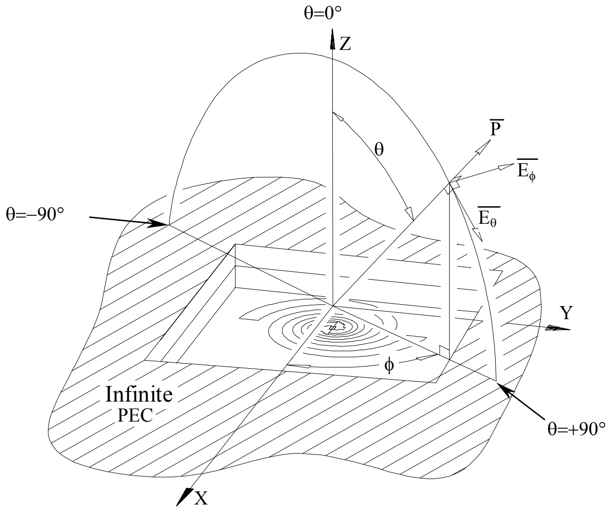
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Total Radiated Power (Sweep Theta): PPC_TPwr

Summary

Also known as a Principal Plane Cut or Theta Sweep. This measurement captures the total power in all polarizations, and fixes the values of Frequency and Phi while sweeping Theta from -90 to 90 degrees or $-\pi/2$ to $\pi/2$ radians. The total power is defined as the sum of the power contained in E_θ and E_ϕ :

$$TPwr = \frac{1}{240\pi}(|E_\theta|^2 + |E_\phi|^2)$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Fixed Phi (degrees)	Real	-90 to 90
Frequency Sweep Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction for all polarizations. This result is normalized to P_{ave} - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave} = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result = \left. \sqrt{\frac{TPwr(\theta, \phi)}{P_{ave}}} \right|_{-90° \le \theta \le 90°,\ \phi = const}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent to a wave variable in the specified direction in order to preserve compatibility with other antenna measurements although the measurement is purely real. This insures that $|\mathrm{result}|^2$ is the directivity in that particular direction. The result can be displayed as a real value by specifying the magnitude, or real component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in angle units.

Note

During P_{ave} computation progress a bar may be displayed warning that a lengthy computation is in progress at the specified frequency. The bar window may display a message “Increased Accuracy Required: dAng=value” which warns that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Axial Ratio (Sweep Frequency): SF_AR

Summary

SF_AR calculates axial ratio. The values of Theta and Phi are fixed while Frequency is swept.

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the axial ratio versus frequency at a fixed theta and phi position. Although axial ratio is a real value, it is returned as a complex value to remain consistent with the other antenna measurements. The axial ratio is defined as the absolute value of the sum of right-hand circular polarization (RHCP) E-fields and left-hand circular polarization (LHCP) E-fields divided by the difference.

Computational Details

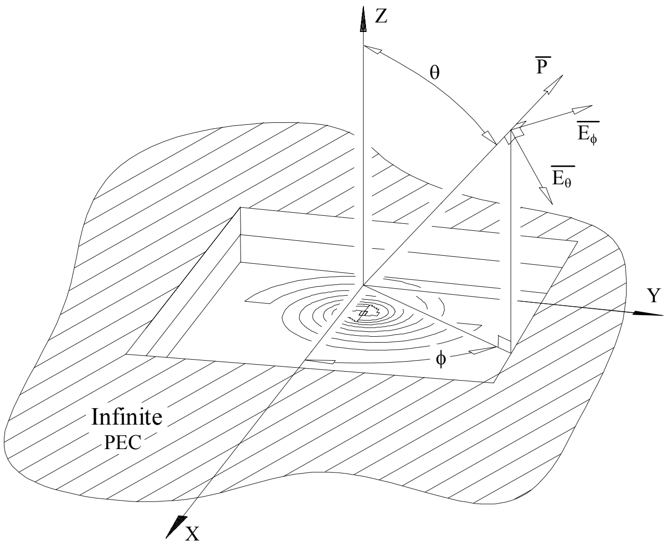
The axial ratio is calculated as:

$$\left|\frac{E_{RHCP} + E_{LHCP}}{E_{RHCP} - E_{LHCP}}\right|$$

E-Phi (Sweep Frequency): SF_EPhi

Summary

Also known as a Swept Frequency measurement polarized along E_ϕ , this measurement fixes the values of Theta and Phi while sweeping Frequency.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave}(F) = \frac{1}{8\pi} Re \int_{0}^{2\pi} \int_{0}^{\pi/2} (E_{\theta} \cdot H^{*}{}_{\phi} - E_{\phi} \cdot H^{*}{}_{\theta}) \sin\theta d\theta d\phi$$

$$result(F) = \left. \frac{E_{\phi}(\theta, \phi, F)}{\sqrt{240\pi P_{ave}(F)}} \right|_{\theta = const, \phi = const} \qquad \text{where F is a frequency}$$

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in frequency units.

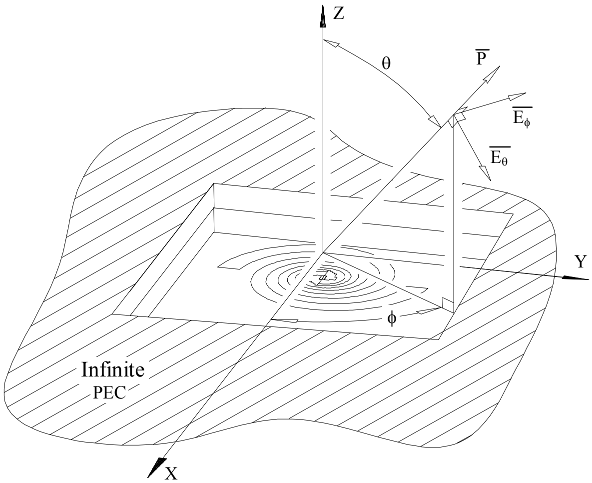
Note

During P_{ave} computation a progress bar may display to warn that a lengthy computation is in progress at the specified frequency. The message “Increased Accuracy Required: dAng=value” warns that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration is reduced by half.

E-Theta (Sweep Frequency): SF_ETheta

Summary

Also known as a Swept Frequency measurement polarized along E_θ, this measurement fixes the values of Theta and Phi while sweeping Frequency.



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave}(F) = \frac{1}{8\pi}Re\int_0^{2\pi}\int_0^{\pi/2}(E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta)\sin\theta d\theta d\phi$$

$$result(F) = \left.\frac{E_\theta(\theta, \phi, F)}{\sqrt{240\pi P_{ave}(F)}}\right|_{\theta = const, \phi = const}$$

where F is a frequency.

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The the result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in frequency units.

Note

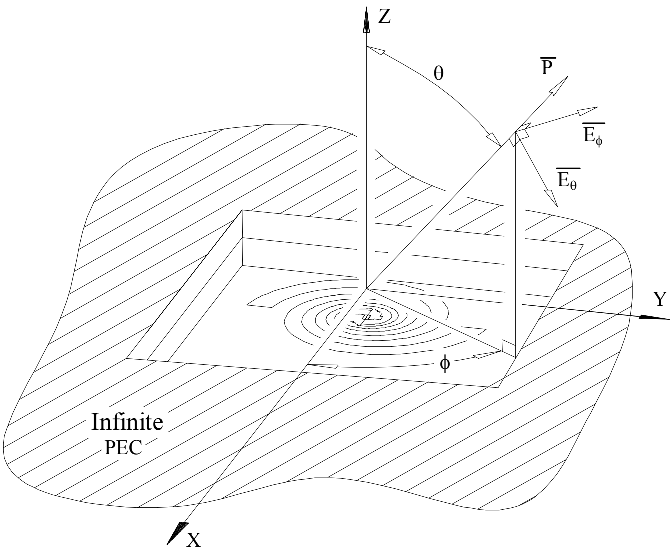
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. This window may also display a message “Increased Accuracy Required: dAng=value.” This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

E-Left-Hand Circular Polarization (Sweep Frequency): SF_LHCP

Summary

Also known as a Frequency Sweep in Left Hand Circular polarization, this measurement fixes the values of Theta and Phi while sweeping frequency. Left Hand Circular polarization is a linear combination of E_θ and E_ϕ as defined below.

$$LHCP(\theta, \phi) = \frac{E_\theta - jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave}(F) = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result(F) = \left. \frac{LHCP(\theta, \phi, F)}{\sqrt{240\pi P_{ave}(F)}} \right|_{\theta = const, \, \phi = const}$$

where F is a frequency.

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allow one to create arrays of these elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in frequency units.

Note

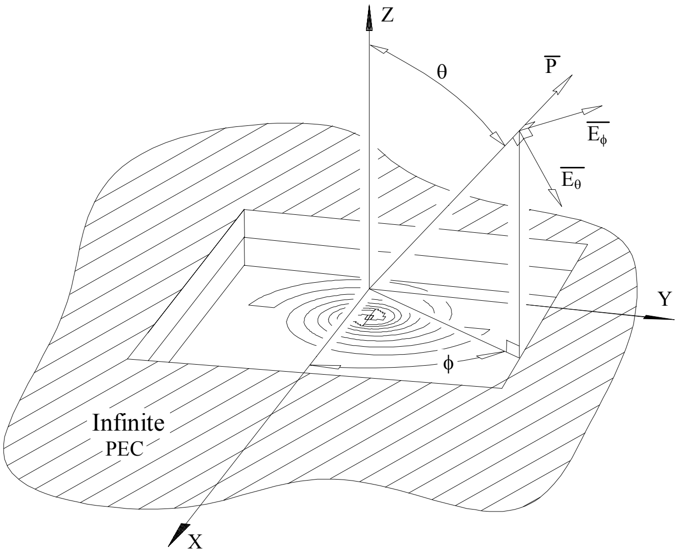
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

E-Right-Hand Circular Polarization (Sweep Frequency): SF_RHCP

Summary

Also known as a Frequency Sweep in Right Hand Circular polarization, this measurement fixes the values of Theta and Phi while sweeping frequency. Right Hand Circular polarization is a linear combination of E_θ and E_ϕ as defined below.

$$RHCP(\theta, \phi) = \frac{E_\theta + jE_\phi}{\sqrt{2}}$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction and polarization. This result is normalized to $\sqrt{P_{ave}}$ - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave}(F) = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta \, d\theta \, d\phi$$

$$result(F) = \left. \frac{RHCP(\theta, \phi, F)}{\sqrt{240\pi P_{ave}(F)}} \right|_{\theta = const,\ \phi = const}$$

where F is a frequency.

The measurement does not reflect the effect of mismatch or resistive losses. The result is an equivalent wave variable in the specified direction, such that $|\text{result}|^2$ is the partial directivity in that particular direction. This allows one to create array radiation patterns of elements by directly adding the complex results of multiple elements. Importantly, the phase center of the measurement is located at the center of the top surface of the enclosure. The result can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in frequency units.

Note

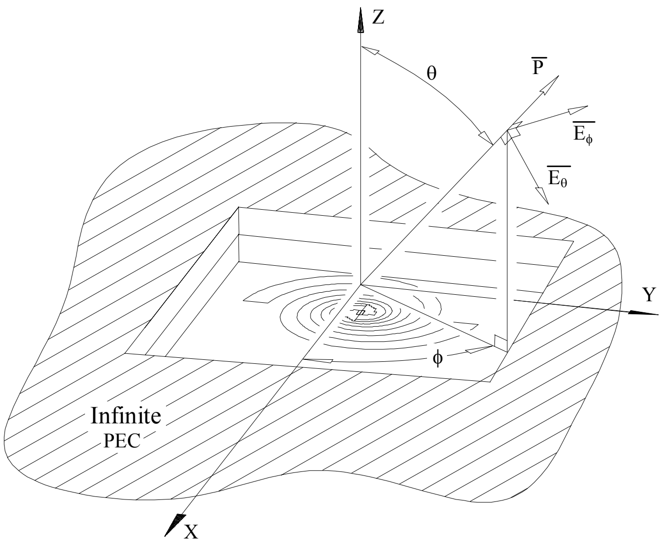
During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Total Radiated Power (Sweep Frequency): SF_TPwr

Summary

Also known as a Frequency Sweep which captures the total power in all polarizations, this measurement fixes the values of Phi and Theta while sweeping frequency. The total power is defined as the sum of the power contained in E_θ and E_ϕ :

$$TPwr = \frac{1}{240\pi}(|E_\theta|^2 + |E_\phi|^2)$$



Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real	0 to 90
Phi (degrees)	Real	-180 to 180

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value which represents the normalized far field radiation in the specified direction for all polarizations. This result is normalized to P_{ave} - an integration of the power (in all polarizations) in the upper hemisphere divided by 4π (asterisk denotes complex conjugate values):

$$P_{ave}(F) = \frac{1}{8\pi} Re \int_0^{2\pi} \int_0^{\pi/2} (E_\theta \cdot H^*_\phi - E_\phi \cdot H^*_\theta) \sin\theta d\theta d\phi$$

$$result(F) = \left. \sqrt{\frac{TPwr(\theta, \phi, F)}{P_{ave}(F)}} \right|_{\theta = const, \phi = const}$$

where F is a frequency.

The measurement does not reflect the effect of mismatch or resistive losses. The result is equivalent wave variable in the specified direction in order to preserve compatibility with other antenna measurements although the measurement is purely real. The $|\text{result}|^2$ is the directivity in the specified direction. The result can be displayed as a real value by specifying the magnitude, or real component in the Add/Modify Measurement dialog box. This value can also be displayed in dB by selecting the **DB** check box which displays 20 log(|result|). The independent axis for this measurement is in frequency units.

Note

During P_{ave} computation progress bar may be displayed warning user that lengthy computation is in progress at the specified frequency. The bar window may display a message "Increased Accuracy Required: dAng=value." This message delivers a warning that the average power is repeatedly computed at increased accuracy and the angular step used in numerical integration was reduced by half.

Generate LVS File: LVS

Summary

LVS generates an LVS netlist for the circuit schematic specified in **Data Source Name**.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

An LVS netlist text file is created in the same directory as the project.

Graph Type

The measurement can be displayed on a rectangular graph.

Generate Netlist: NETDMP

Summary

NETDMP generates an AWR netlist for the circuit schematic specified in **Data Source Name**.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

An AWR netlist text file is created in the same directory as the project.

Graph Type

This measurement is plotted on a rectangular graph.

Auxiliary Stability Factor: B1

Summary

B1 is the supplemental stability factor for a two port, defined as:

$$B1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta|^2$$

where Δ was defined above for K. The necessary and sufficient conditions for unconditional stability are:

$K > 1$ and $B1 > 0$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Capacitance of Input as a Parallel RC: C_PRC

Summary

C_PRC can be used to compute the capacitance value of a parallel resistor/capacitor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in capacitance units.

Capacitance of Input as a Series RC: C_SRC

Summary

C_SRC can be used to compute the capacitance value of a series resistor/capacitor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in capacitance units.

Computational Details

The capacitance is calculated as follows:

$$C = \frac{-1}{\omega \cdot \mathrm{imag}(Z)}$$

where Z is the calculated complex impedance looking into the specified port and ω is the angular frequency ($\omega = 2 \cdot \pi \cdot f$ where f is the simulation frequency).

Gamma1 Measured With Gamma-Probe: GAM1_GP

Summary

GAM1_GP is used for the calculation of internal reflection coefficient Γ_1 in conjunction with the Gamma-Probe element. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	3 to 1000 ports
Excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Zo, Real value (Ohms)	Real	0.1 to 2500
Zo, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Gamma2 Measured With Gamma-Probe: GAM2_GP

Summary

GAM2_GP is used for the calculation of internal reflection coefficient Γ_2 in conjunction with the Gamma-Probe element. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	3 to 1000 ports
Excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Zo, Real value (Ohms)	Real	0.1 to 2500
Zo, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real measurement can also be displayed in dB by selecting the **DB** check box.

Gamma measured with modified Gamma-Probe: GAM_GPM

Summary

GAM_GPM is used for the calculation of internal reflection coefficient in conjunction with the Modified Gamma-Probe element. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element and the AWR Knowledge Base for more information on the difference between the regular and modified gamma probe.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	3 to 1000 ports
Excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Voltage Sample Port On Node 1 Side	Integer	1 to 1000
Voltage Sample Port On Node 2 Side	Integer	1 to 1000
Gamma Selection	List of options	Gamma 1 or Gamma 2
Zo, Real value (Ohms)	Real	0.1 to 2500
Zo, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real measurement can also be displayed in dB by selecting the **DB** check box.

Group Delay: GD

Summary

GD is used to compute the group delay between any 2-ports of an N-port network. This measurement first unwraps the phase of the argument using 90-degrees as the trigger to unwrap. Alternate triggers can be used by using the 'unwrap' function and writing equations. The group delay is calculated from

$$GD = -\left.\frac{d\varphi_{ij}(\omega)}{d\omega}\right|_{\omega = \omega_0}$$

where $\varphi_{ij}(\omega_{\omega_0})$ is defined from the S parameters written as

$$S_{ij} = |S_{ij}|e^{j\varphi_{ij}(\omega)}$$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

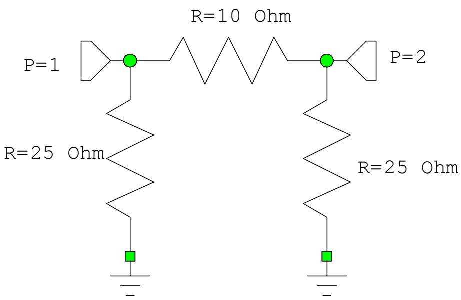
Result

This measurement returns a real value in time units.

Even Mode Gamma: Geven

Summary

The even-mode gamma returns the reflection coefficient looking into one of a pair of ports that are driven by an even-mode excitation. An *even-mode excitation* means that two equal, in-phase sources are connected to a pair of terminals. The reflection coefficient is computed from the voltage and current at one of the ports, the one designated as “First Port” in the measurement set-up.



In the example shown above, the even-mode gamma using ports 1 and 2 is computed from the even mode impedance of 25 Ohms. Normally this measurement is used with circuits that are symmetric with respect to the nodes, and Geven[1,2] (i.e., 1 is the first port and 2 is the second port) then equals Geven[2,1]. If the circuit is not symmetric with respect to the excited nodes, Geven[1,2] does not equal Geven[2,1].

The reflection coefficient is presented as a reflection coefficient in a 50 Ohm system.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
First port	Integer	1 to 1000
Second port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Simultaneous Match at Input: GM1

Summary

GM1 returns the reflection coefficient that must be seen by the input to achieve a simultaneous conjugate match at both the input and output. The reflection coefficient is presented as a reflection coefficient in a 50 Ohm system. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Simultaneous Match at Output: GM2

Summary

GM2 returns the reflection coefficient that must be seen by the output to achieve a simultaneous conjugate match at both the input and output. The reflection coefficient is presented as a reflection coefficient in a 50 Ohm system. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

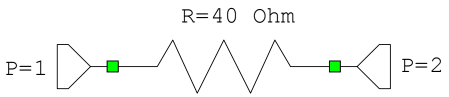
Result

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 real value can also be displayed in dB by selecting the **DB** check box.

Odd Mode Gamma: Godd

Summary

The odd-mode gamma returns the reflection coefficient looking into one of a pair of ports that are driven by an odd-mode excitation. An *odd-mode excitation* means that two equal magnitude sources, 180 degrees out of phase, are connected to a pair of terminals. The reflection coefficient is computed from the voltage and current at one of the ports, the one designated as "First Port" in the measurement set-up.



In the example shown above, the odd-mode gamma using ports 1 and 2 is computed from the odd-mode impedance of 20 Ohms. The impedance is computed from V/I where +V is applied to port 1 and -V is applied to port 2, and I is the current in the specified port. Another way to view this is that the odd mode excitation establishes a virtual ground half way between the two ports, so the circuit appears as two 20 Ohm resistors in series, with the center connection point a virtual ground.

Normally this measurement is used with circuits that are symmetric with respect to the excited nodes and Godd[1,2] (i.e., 1 is the first port and 2 is the second port) then equals Godd[2,1]. If the circuit is not symmetric with respect to the 2 nodes, Godd[1,2] does not equal Godd[2,1].

The reflection coefficient is presented as a reflection coefficient in a 50 Ohm system.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
First port	Integer	1 to 1000
Second port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter

Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Stability Factor: K

Summary

K is the stability factor for a two port, defined as:

$$K = \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |\Delta|^2}{2|S_{12}S_{21}|}$$

where

$$\Delta = S_{11}S_{22} - S_{12}S_{21}$$

The necessary and sufficient conditions for unconditional stability are:

$$K > 1 \text{ and } B1 > 0$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Inductance of Input as a Parallel RL: L_PRL

Summary

L_PRL can be used to compute the inductance value of a parallel resistor/inductor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in inductance units.

Computational Details

The inductance is calculated as follows:

$$L = \frac{\mathrm{real}(Z)^2 + \mathrm{imag}(Z)^2}{\omega \cdot \mathrm{imag}(Z)}$$

where Z is the calculated complex impedance looking into the specified port and ω is the angular frequency ($\omega = 2 \cdot \pi \cdot f$ where f is the simulation frequency).

Inductance of Input as a Series RL: L_SRL

Summary

L_SRL can be used to compute the inductance value of a series resistor/inductor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the Microwave Office/Analog Office User Guide for details on configuring these parameters.

Result

This measurement returns a real value in inductance units.

Linear Deviation from Phase: LDVP

Summary

LDVP measures the deviation of the phase from the linear phase.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1-1000 Ports
To Port Index	Integer	1-1000
From Port Index	Integer	1-1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the value in Degrees. This measurement can be displayed on a rectangular or tabular graph.

Implementation Detail

The measurement unwraps the phase of S(i,j) using a 90 degree threshold. It then fits a straight line to this unwrapped phase and computes the difference between the line and the unwrapped phase.

Geometric Stability Factor (Load): MU1

Summary

MU1 computes the geometric stability factor of a 2-port. The geometric stability factor computes the distance from the center of the Smith chart to the nearest unstable point of the output load plane. The necessary and sufficient condition for unconditional stability of the two port is that MU1 > 1. The stability factor is computed from

$$MU1 = \frac{1 - |S_{11}|^2}{|S_{22} - S_{11}^*\Delta| + |S_{21}S_{12}|}$$

where

$$\Delta = S_{11}S_{22} - S_{12}S_{21}$$

and * indicates the complex conjugate.

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box

Geometric Stability Factor (Source): MU2

Summary

MU2 computes the geometric stability factor of a 2-port. The geometric stability factor computes the distance from the center of the Smith chart to the nearest unstable point of the input source plane. The necessary and sufficient condition for unconditional stability of the two port is that MU2 > 1. The stability factor is computed from

$$MU2 = \frac{1 - |S_{22}|^2}{|S_{11} - S_{22}^{*}\Delta| + |S_{21}S_{12}|}$$

where

$$\Delta = S_{11}S_{22} - S_{12}S_{21}$$

and * indicates the complex conjugate.

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Passive: PASSIVE

Summary

The PASSIVE measurement is designed to help determine if a subcircuit represented by its frequency-dependent S-parameters is passive (i.e. that it does not generate power). This measurement is useful for the subcircuits represented by Touchstone files or EM structures.

This measurement calculates the smallest eigenvalue of the matrix $A=U-S^H S$, where U is the identity matrix, S is the scattering matrix, and the superscript "H" denotes Hermitian conjugate. For the S matrix to be passive, the smallest eigenvalue of the matrix A defined above should be non-negative. This calculation is performed for each frequency in the sweep.

This passivity criterion is both necessary and sufficient, while the requirement that the magnitude of Sij should not exceed 1 is just necessary.[1]

If the result is non-negative, the circuit is passive. If it is negative, it is not passive (generates power).

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Simulator	String	Default Linear, HSPICE Linear, Spectre Linear

NOTE. All measurements have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The result is a real value that is positive or zero at frequencies where the subcircuit is passive, and negative at the frequencies where it is non-passive. Due to experimental errors and the errors in EM simulations, small negative values result even for passive structures. The rule of thumb for the tolerance is that for EM simulations, negative values should not exceed 10^{-5} in magnitude, and 10^{-4} for experimental results for the subcircuit to be considered passive.

The results should not be plotted in dB as the sign can be negative, and converting to dB causes this information to be lost.

Computational Details

The assumption is that Re $Z_0>0$ where Z_0 is the characteristic impedance. This assumption is almost always valid in practice.

References

[1] K. Kurokawa, Power Waves and The Scattering matrix, MTT, March 1964.

Resistance of Input as a Parallel RC: R_PRC

Summary

R_PRC can be used to compute the resistance value of a parallel resistor/capacitor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subciruit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in resistance units.

Resistance of Input as a Parallel RL: R_PRL

Summary

R_PRL can be used to compute the resistance value of a parallel resistor/inductor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in resistance units.

Computational Details

The resistance is calculated as follows:

$$R = \frac{\mathrm{real}(Z)^2 + \mathrm{imag}(Z)^2}{\mathrm{real}(Z)}$$

where Z is the calculated complex impedance looking into the specified port and ω is the angular frequency ($\omega = 2 \cdot \pi \cdot f$ where f is the simulation frequency).

Resistance of Input as a Series RC: R_SRC

Summary

R_SRC can be used to compute the resistance value of a series resistor/capacitor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in resistance units.

Computational Details

The resistance is calculated as follows:

$$R = \mathrm{real}(Z)$$

where Z is the calculated complex impedance looking into the specified port and ω is the angular frequency ($\omega = 2 \cdot \pi \cdot f$ where f is the simulation frequency).

Resistance of Input as a Series RL: R_SRL

Summary

R_SRL can be used to compute the resistance value of a series resistor/inductor that has the same impedance as the impedance looking into the specified port. All other ports are terminated using the impedances specified by the port terminations.

Parameters

Name	Type	Range
Data Source Name	Subciruit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in resistance units.

Stability Index Measured With Gamma-Probe: STAB_GP

Summary

STAB_GP plots the stability index as a function of frequency. It is used in conjunction with the Gamma-Probe element. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element.

Parameters

Name	Type	Range
Data Source Name	Subciruit	4 to 1000
Gamma1 excitation port	Integer	1 to 1000
Gamma2 excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Zo1, Real value (Ohms)	Real	0.1 to 2500
Zo1, imag (Ohms)	Real	-2500 to 2500
Zo2, Real value (Ohms)	Real	0.1 to 2500
Zo2, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real measurement can also be displayed in dB by selecting the **DB** check box.

Stability index measured with modified Gamma-Probe: STAB_GPM

Summary

STAB_GPM plots the stability index as a function of frequency. It is used in conjunction with the Modified Gamma-Probe element. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element and the AWR Knowledge Base for more information on the difference between the regular and modified gamma probe.

Parameters

Name	Type	Range
Data Source Name	Subciruit	4 to 1000
Gamma1 excitation port	Integer	1 to 1000
Gamma2 excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Voltage Sample Port On Node 1 Side	Integer	1 to 1000
Voltage Sample Port On Node 2 Side	Integer	1 to 1000
Zo1, Real value (Ohms)	Real	0.1 to 2500
Zo1, imag (Ohms)	Real	-2500 to 2500
Zo2, Real value (Ohms)	Real	0.1 to 2500
Zo2, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real measurement can also be displayed in dB by selecting the **DB** check box.

Nyquist Stability Measured With Gamma-Probe: STAB_GPN

Summary

STAB_GPN is used for plotting the open-loop gain function in conjunction with the Gamma-Probe element. The plots are useful for examination of circuit stability by application of the Nyquist criterion. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element and the STAB_GPN measurement.

Parameters

Name	Type	Range
Data Source Name	Subciruit	4 to 1000
Gamma1 excitation port	Integer	1 to 1000
Gamma2 excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Zo1, Real value (Ohms)	Real	0.1 to 2500
Zo1, imag (Ohms)	Real	-2500 to 2500
Zo2, Real value (Ohms)	Real	0.1 to 2500
Zo2, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Nyquist Stability Measured with Modified Gamma-Probe: STABN_GPM

Summary

STABN_GPM is used for plotting the open-loop gain function in conjunction with the Gamma-Probe element. The plots are useful for examination of circuit stability by application of the Nyquist criterion. See the chapter on internal stability analysis in the Reference Guide for details of use of the Gamma-Probe element and the STABN_GPM measurement

Parameters

Name	Type	Range
Data Source Name	Subciruit	4 to 1000
Gamma1 excitation port	Integer	1 to 1000
Gamma2 excitation port	Integer	1 to 1000
Voltage sample port	Integer	1 to 1000
Current sample port	Integer	1 to 1000
Voltage Sample Port On Node 1 Side	Integer	1 to 1000
Voltage Sample Port On Node 2 Side	Integer	1 to 1000
Zo1, Real value (Ohms)	Real	0.1 to 2500
Zo1, imag (Ohms)	Real	-2500 to 2500
Zo2, Real value (Ohms)	Real	0.1 to 2500
Zo2, imag (Ohms)	Real	-2500 to 2500

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real measurement can also be displayed in dB by selecting the **DB** check box.

Summation of Power in Network: SUMPWR

Summary

SUMPWR is used to calculate the total, relative power accounted for in a linear, passive network.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Excitation Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unitless real value. The measurement can be displayed in dB by selecting the **DB** check box in the Add/Modify Measurement dialog box.

Computational Details

Using the calculated s-parameters of the circuit, the conservation of power law is used to calculate the total power:

$$\sum_{i=1}^{N} |S_{Ni}|^2 = 1$$

The value returned by this measurement will be between 0 and 1 for a linear, passive network.

As an example, if the value returned by the measurement is 0.95, this indicates that 95% of the total power can be accounted for at all ports in the circuit. 5% of the total power would be either in the form of resistive losses or stored (complex) power in the circuit.

Voltage Standing Wave Ratio: VSWR

Summary

VSWR is the Voltage Standing Wave Ratio at a port with all other ports terminated in with the specified port terminations. The VSWR is defined as:

$$VSWR = \frac{1 + |\Gamma_0|}{1 - |\Gamma_0|}$$

where Γ_o is defined as the reflection coefficient at the port with all other ports terminated.

Parameters

Name	Type	Range
Data Source Name	Subciruit	1 to 1000 ports
Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

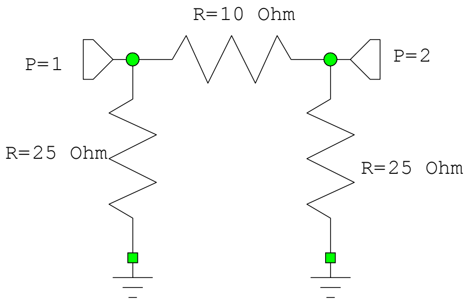
Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Even Mode Admittance: Yeven

Summary

The even-mode admittance returns the admittance looking into one of a pair of ports that are driven with an even-mode excitation. An *even-mode excitation* means that two equal, in-phase sources are connected to a pair of terminals. The admittance is computed from the current in the terminals and the applied in-phase voltages.



In the example shown above, the even-mode admittance using ports 1 and 2 is 0.04 S (i.e., 25Ω). Note that the mode admittance is defined as the admittance looking into one port; in this case, it is the port designated as "First Port" in the measurement set-up.

Normally this measurement is used with circuits that are symmetric with respect to the excited nodes and Yeven[1,2] (i.e., 1 is the first port and 2 is the second port) is the same as Yeven[2,1]. If the circuit is not symmetric with respect to the excited nodes, Yeven[1,2] does not equal Yeven[2,1].

Parameters

Name	Type	Range
Data Source Name	Subciruit	2 to 1000 ports
First port	Integer	1 to 1000
Second port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

Input Admittance at a Port: YIN

Summary

YIN computes the input admittance looking into a port with all other ports terminated using the impedances specified by the port terminations. Note, this would not be the same as Y_{ii} using y-parameters since y-parameters terminate all other ports with short circuits. The only case where they will match will be for a one port network.

Parameters

Name	Type	Range
Data Source Name	Subciruit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

Simultaneous Admittance Match at Input: YM1

Summary

YM1 returns the admittance that must be seen by the input to achieve a simultaneous conjugate match at both the input and output. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Simultaneous Admittance Match at Output: YM2

Summary

YM2 returns the admittance that must be seen by the output to achieve a simultaneous conjugate match at both the input and output. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

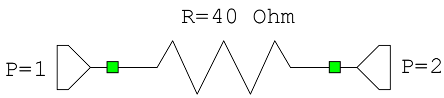
Result

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 real value can also be displayed in dB by selecting the **DB** check box.

Odd Mode Admittance: Yodd

Summary

The odd-mode admittance returns the admittance looking into one of a pair of ports that are driven with an odd-mode excitation. An o*dd-mode excitation* means that two equal-magnitude sources, are 180 degrees out of phase, are connected to a pair of terminals. The admittance is computed from the voltage and current at one of the ports, the one designated as "First Port" in the measurement set-up.



In the example shown above, the odd-mode admittance using ports 1 and 2 is 0.05 S (i.e., 20Ω). The admittance is computed from I/V where +V is applied to port 1 and –V is applied to port 2. Another way to view this is that the odd mode excitation establishes a virtual ground half way between the two ports, so the circuit appears as two 20 Ohm resistors in series, with the center connection point a virtual ground.

Normally this measurement is used with circuits that are symmetric with respect to the excited nodes and Yodd[1,2] (i.e., 1 is the first port and 2 is the second port) is the same as Yodd[2,1]. If the circuit is not symmetric with respect to the nodes, Yodd[1,2] does not equal Yodd[2,1].

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000
First port	Integer	1 to 1000
Second port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

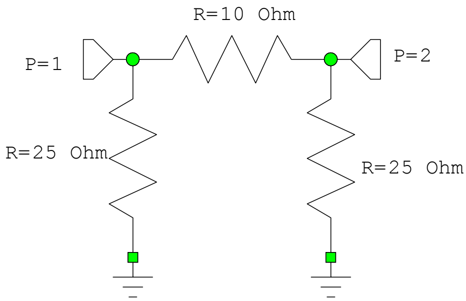
Result

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.

Even Mode Impedance: Zeven

Summary

The even-mode impedance returns the impedance looking into one of a pair of ports that are driven with an even-mode excitation. An *even-mode excitation* means that two equal, in-phase sources are connected to a pair of terminals. The impedance is computed from the current in the terminals and the applied in-phase voltages.



In the example shown above, the even mode impedance using ports 1 and 2 is 25 Ohms. Note that the mode impedance is defined as the impedance looking into one port; in this case, it is the port listed as "First Port" in the measurement set-up.

Normally this measurement is used with circuits that are symmetric with respect to the nodes that are excited in-phase. Then, Zeven[1,2] (i.e., 1 is the first port and 2 is the second port) is the same as Zeven[2,1]. If the circuit is not symmetric with respect to the excited nodes, Zeven[1,2] does not equal Zeven[2,1].

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000
First port	Integer index	1 to 1000
Second port	Integer index	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter

Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

Input Impedance at a Port: ZIN

Summary

ZIN computes the input impedance looking into a port with all other ports terminated using the impedances specified by the port terminations. Note, this would not be the same as Z_{ii} using z-parameters since z-parameters terminate all other ports with open circuits. The only case where they will match will be for a one port network.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Port index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

Simultaneous Impedance Match at Input: ZM1

Summary

ZM1 returns the impedance that must be seen by the input to achieve a simultaneous conjugate match at both the input and output. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

Simultaneous Impedance Match at Output: ZM2

Summary

ZM2 returns the impedance that must be seen by the output to achieve a simultaneous conjugate match at both the input and output. This measurement is not dependent on the port termination impedance values that can be specified in the circuit. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

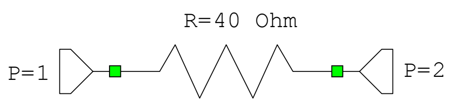
Result

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.

Odd Mode Impedance: Zodd

Summary

The odd-mode impedance returns the impedance looking into one of a pair of ports that are driven with an odd-mode excitation. An *odd-mode excitation* means that two equal-magnitude sources, 180 degrees out of phase, are connected to a pair of terminals. The voltages at the ports are +I and –I. The impedance is computed from the current in the specified terminal and the applied out-of-phase voltages.



In the example shown above, the odd-mode impedance using ports 1 and 2 is 20 Ohms. The impedance is computed from V/I where +I is applied to port 1 and –I is applied to port 2, and I is the current in the specified port. (V/I is calculated at the port designated as “First Port” in the measurement set-up.) Another way to view this is that the odd mode excitation establishes a virtual ground half way between the two ports, so the circuit appears as two 20 Ohm resistors in series, with the center connection point being a virtual ground.

Normally, this measurement is used with circuits that are symmetric with respect to the excited nodes; then Zodd[1,2] is the same as Zodd[2,1]. If the circuit is not symmetric with respect to the nodes, Zodd[1,2] does not equal Zodd[2,1].

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
First port	Integer	1 to 1000
Second port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter

Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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.

AC Current: Iac

Summary

Iac measures AC current at a point in the circuit specified by the **Measurement Component** parameter. This measurement is only applied to HSPICE AC and APLAC AC analysis.

If the **Measurement Component** specifies a node, then the current measured is the current entering this node. If the **Measurement Component** specifies just an element, the current is through the element. The current value is returned as the complex magnitude.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in current units. 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 real value can also be displayed in dB by selecting the **dB** checkbox.

Differential AC Current:IacD

Summary

IacD measurse AC current between two points in the circuit as specified by the **+Measurement Component** and **-Measurement Component** parameters. This measurement is only applied to the HSPICE AC and APLAC AC analysis.

If **Measurement Component** specifies a node, the current measured is the current entering this node. If **Measurement Component** specifies just an element, the current is through the element. The current value is returned as the complex magnitude.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component+	String	N/A
Measurement Component-	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in current units. 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 real value can also be displayed in dB by selecting the **dB** checkbox.

Output Noise Voltage: NoiseC

Summary

NoiseC measures the output AC noise voltage due to a noise contributor specified in **Noise Contributor Name**. The noise is measured using a noise meter V_NSMTR specified in **Output Noise Meter**. This measurement is only applied to HSPICE AC and APLAC AC analysis.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output Noise Meter	String	N/A
Noise Contributor Name	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in V^2/Hz.

AC Noise Contributors: NoiseCon

Summary

NoiseCon lists the elements and the amount of noise they contibute to the noise measured in a circuit using a noise meter (V_NSMTR) specified in **Output Noise Meter**. The **Noise Type** is either **Total Noise** or **All Noise**. The result is sorted by choosing a **Sort Criterion**. This measurement is only applied to HSPICE AC and APLAC AC analysis.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output Noise Meter	String	N/A
Noise Type	Real	N/A
Sort Criterion	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

The measurement is plotted in a Tabular graph.

Equivalent Input Noise Voltage: NoiseI

Summary

NoiseI measures the equivalent input AC noise voltage using a noise meter (V_NSMTR) specified in **Output Noise Meter**. This measurement is only applied to HSPICE AC and APLAC AC analysis.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output Noise Meter	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in V^2/Hz.

Total Output Noise Voltage: NoiseO

Summary

NoiseO measures the total output AC noise voltage using a noise meter (V_NSMTR) specified in **Output Noise Meter**. This measurement is only applied to HSPICE AC and APLAC AC analysis.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output Noise Meter	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in V^2/Hz.

AC Voltage: Vac

Summary

Vac measures the AC voltage measured at a point in the circuit as specified by the **Measurement Component** parameter. This measurement is only applied to HSPICE AC and APLAC AC analysis.

If the **Measurement Component** parameter specifies a node, then the voltage measured is the voltage at this node referenced to ground. If the **Measurement Component** parameter specifies an element with two nodes, the voltage measured is the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error is generated. The voltage value is returned as the complex magnitude of the voltage

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. You can display the complex measurement as a real value by specifying the magnitude (**Mag.**), **Angle**, **Real** or imaginary (**Imag.**) component in the Add/Modify Measurement dialog box. You can also display the real value dB by selecting the **dB** checkbox which then displays 20*log10(|Val|).

Differential AC Voltage: VacD

Summary

VacD measures the AC voltage between two points in the circuit as specified by the **+Measurement Component** and **-Measurement Component** parameters. This measurement is only applied to HSPICE AC and APLAC AC analysis.

If **Measurement Component** specifies a node, then the voltage used is the voltage at this node referenced to ground. If **Measurement Component** specifies an element with two nodes, the voltage used is the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error is generated. The voltage value is returned as the complex magnitude of the voltage.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component+	String	N/A
Measurement Component-	String	N/a

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. You can display the complex measurement as a real value by specifying the magnitude (**Mag.**), **Angle**, **Real** or imaginary (**Imag.**) component in the Add/Modify Measurement dialog box. You can also display the real value in dB by selecting the **dB** checkbox which then displays 20*log10(|Val|).

Available Gain Circles: GACIR

Summary

GACIR displays constant available gain contours in the input reflection plane. The gain indicated by each contour is specified by selecting a maximum gain for the first contour and the gain step between the rest of the contours. For more information on the definition of available gain, see the documentation for the GA measurement. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only
Max gain (db)	Real	-200 to 200
Gain step	Real	0 to 200
Number circles	Integer	1 to 20

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Available Gain Circles Starting at Maximum Available Gain: GAC_MAX

Summary

GAC_MAX displays constant available gain contours in the input reflection plane. This measurement uses the value of GMAX for the first contour, with a specified gain step between the rest of the contours. For more information on the definition of available gain, see the documentation for the GA measurement. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only
Gain step	Real	0 to 200
Number circles	Integer	1 to 20

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Power Gain Circles: GPCIR

Summary

GPCIR displays constant power gain contours in the output reflection plane. The gain indicated by each contour is specified by selecting a maximum gain for the first contour and the gain step between the rest of the contours. For more information on the definition of power gain, see the documentation for the GP measurement. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only
Max gain (db)	Real	-200 to 200
Gain step	Real	0 to 200
Number circles	Integer	1 to 20

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Power Gain Circles Starting at Maximum Power Gain: GPC_MAX

Summary

GPC_MAX displays constant power gain contours in the output reflection plane. This measurement uses the value of GMax for the first contour, with a specified gain step between the rest of the contours. For more information on the definition of power gain, see the documentation for the GP measurement. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only
Gain step	Real	0 to 200
Number circles	Integer	1 to 20

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Constant Impedance Mismatch Circle: MMCIRC

Summary

MMCIRC is used to display a contour in the impedance plane which provides a constant mismatch using a specified port. The mismatch is specified as return loss. The mismatch contour will be presented on a 50 Ohm normalized Smith chart and all ports other than the one being mismatched will be terminated using the terminations specified in the circuit.

Parameters

Name	Type	Range
Source Name	Subcircuit	1 to 1000 ports
Port to mismatch	Integer	1 to 1000
Return loss (dB)	Real	0 to 200

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Noise Figure Circles: NFCIR

Summary

NFCIR is used to display contours in the source plane which provide a constant noise figure for the device. The value of F_{min} will be the value of the noise figure at the center of the first contour. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only
Number of circles	Integer	1 to 10
Step between (dB)	Real	0 to 100

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Stability Circle at Port I for Gamma=1 at Port J: SCIR_IJ

Summary

SCIR_IJ is used to display stability between any two ports of an N-port. The stability circle is a contour in the I plane that indicates termination values that will make the J plane reflection coefficient values have a unity magnitude. All other ports in the circuit will be terminated using the termination values specified in the circuit. A reflection coefficient less than unity will indicate a stable device, while a reflection coefficient greater than unity indicates a potentially unstable device. The display of the stability circle indicates the unstable region using a circle drawn with a dashed line in the unstable region. If the dashed circle is inside the solid circle, then the outside of the circle indicates the stable region, while if the dashed circle is outside the solid circle, then the inside of the circle represents the stable region.

Parameters

Name	Type	Range
Source Name	Subcircuit	2 to 1000 ports
Stability circle port	Integer	1 to 1000
Gamma|=1 port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Input Stability Circles: SCIR1

Summary

SCIR1 is used to display input stability circles on a Smith chart. The input stability circle is a contour in the source plane that indicates source termination values that will make the output reflection coefficient have a unity magnitude. An output reflection coefficient less than unity will indicate a stable device, while an output reflection coefficient greater than unity indicates a potentially unstable device. The display of the stability circle indicates the unstable region using a circle drawn with a dashed line in the unstable region. If the dashed circle is inside the solid circle, then the outside of the circle indicates the stable region, while if the dashed circle is outside the solid circle, then the inside of the circle represents the stable region. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Output Stability Circles: SCIR2

Summary

SCIR2 is used to display output stability circles on a Smith chart. The output stability circle is a contour in the load plane that indicates load termination values that will make the input reflection coefficient have a unity magnitude. An input reflection coefficient less than unity will indicate a stable device, while an input reflection coefficient greater than unity indicates a potentially unstable device. The display of the stability circle indicates the unstable region using a circle drawn with a dashed line in the unstable region. If the dashed circle is inside the solid circle, then the outside of the circle indicates the stable region, while if the dashed circle is outside the solid circle, then the inside of the circle represents the stable region. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

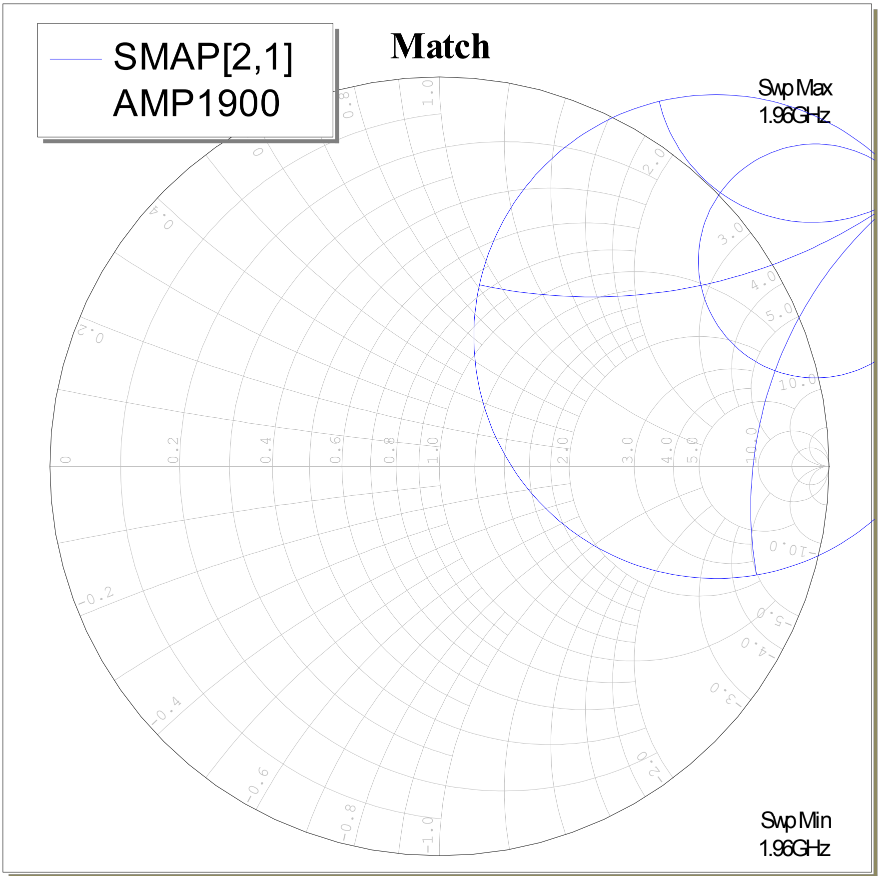
Result

This measurement returns the parameters of a circle or set of circles.

Source Mapping Circles: SMAP

Summary

SMAP is used to display a contour of impedances at the "To" port that results from a contour of impedances at the "From" port. The resulting measurement displays a distorted Smith chart of "From" impedances in the "To" port reflection plane. The diagram below displays an example, where the main Smith chart represents the values presented at the "To" port as a result of any value chosen on the smaller Smith chart at the "From" port.



Parameters

Name	Type	Range
Source Name	Subcircuit	2 to 1000 ports
Map to port	Integer	1 to 1000
Map from port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns the parameters of a circle or set of circles.

Available Gain: GA

Summary

The available gain (also known as the available power gain) is the ratio of the power available from the network to the power available from the source. The available gain is given by

$$G_A = \frac{P_{\text{Available from the network}}}{P_{\text{Available from the source}}}$$

The available gain is computed from

$$G_A = \frac{1 - |\Gamma_S|^2}{|1 - S_{11}\Gamma_s|^2}|S_{21}|^2 \frac{1}{1 - |\Gamma_{\text{out}}|^2}$$

where Γ_{out} , the reflection coefficient looking into the output, is given by

$$\Gamma_{\text{out}} = S_{22} + \frac{S_{12}S_{21}\Gamma_s}{1 - S_{11}\Gamma_s}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Maximum Available Gain: GMax

Summary

For an unconditionally stable two port, GMax is the maximum transducer power gain given as:

$$GMax = \frac{|S_{21}|}{|S_{12}|}(K - \sqrt{K^2 - 1})$$

where K is defined in the documentation for the K measurement. For a two port that is not unconditionally stable, GMax will be defined as the maximum stable gain given as

$$GMax = \frac{|S_{21}|}{|S_{12}|}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

Operating Power Gain: GP

Summary

The operating power gain (also known as the power gain) is the ratio of the power delivered to the load to the power input from the source. The power gain is given by

$$G_p = \frac{P_{\text{Power delivered to the load}}}{P_{\text{Power input to the network}}}$$

The power gain is computed from

$$G_p = \frac{1}{1-|\Gamma_{in}|^2}|S_{21}|^2\frac{1-|\Gamma_L|^2}{|1-S_{22}\Gamma_L|^2}$$

where Γ_{in}, the reflection coefficient looking into the input, is given by

$$\Gamma_{in} = S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1-S_{22}\Gamma_L}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result:

This measurement returns a real value.The real value can also be displayed in dB by selecting the **DB** check box.

Transducer Power Gain: GT

Summary

The transducer power gain is the ratio of the power delivered to the load to the power available from the source. The transducer power gain is given by

$$G_T = \frac{P_{\text{Power delivered to the load}}}{P_{\text{Power available from the source}}}$$

The transducer power gain is computed from

$$G_T = \frac{1-|\Gamma_s|^2}{|1-\Gamma_{in}\Gamma_s|^2}|S_{21}|^2\frac{1-|\Gamma_L^2|^2}{|1-S_{22}\Gamma_L|^2}$$

where Γ_{in}, the reflection coefficient looking into the input, is given by

$$\Gamma_{in} = S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1-S_{22}\Gamma_L}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

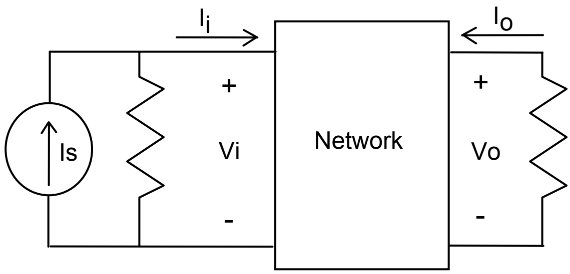
Current Gain from Input Current Source: ISG

Summary

The current gain from the input current source is defined as

$$ISG = \frac{I_o}{I_s}$$

where the currents are defined as shown below.



This measurement can be used to compute the gain between any two ports of an N-port network. For networks with more than two ports, all ports that are not measurement ports are terminated with the termination impedances specified in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

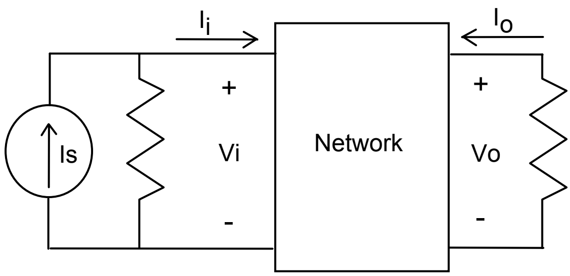
Current Gain from Input Terminal: ITG

Summary

The current gain from the input terminal is defined as

$$ITG = \frac{I_o}{I_i}$$

where the currents are defined as shown below.



This measurement can be used to compute the gain between any two ports of an N-port network. For networks with more than two ports, all ports that are not measurement ports are terminated with the termination impedances specified in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Maximum Stable Gain: MSG

Summary

The maximum stable gain is the maximum gain that can be achieved by a potentially unstable device. Maximum stable gain is given as:

$$MSG = \frac{|S_{21}|}{|S_{12}|}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **DB** check box.

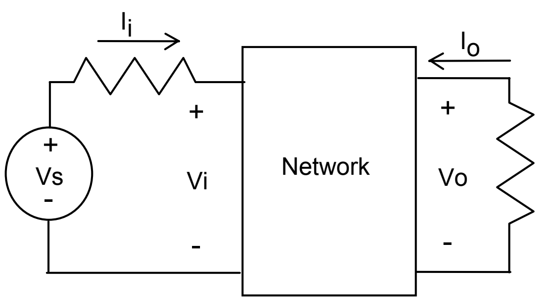
Voltage Gain from Input Voltage Source: VSG

Summary

The voltage gain from the input voltage source is defined as

$$VSG = \frac{V_o}{V_s}$$

where the voltages are defined as shown below.



This measurement can be used to compute the gain between any two ports of an N-port network. For networks with more than two ports, all ports that are not measurement ports are terminated with the termination impedances specified in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

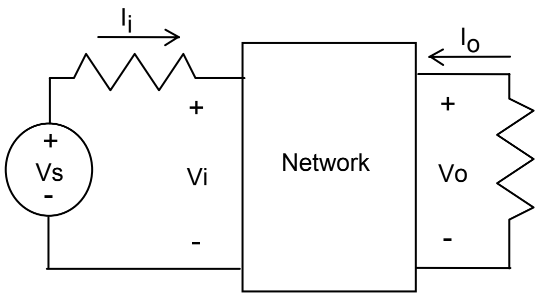
Voltage Gain from Input Terminal: VTG

Summary

The voltage gain from the input terminal is defined as

$$VTG = \frac{V_o}{V_i}$$

where the voltages are defined as shown below.



This measurement can be used to compute the gain between any two ports of an N-port network. For networks with more than two ports, all ports that are not measurement ports are terminated with the termination impedances specified in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	2 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Noise Correlation Matrix: Ci

Summary

Ci computes the N-port current correlation matrix of a linear (or DC-linearized nonlinear) network. The correlation matrix is normalized to $4kT_0$, where T_0=290K.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
To Port Index	Integer	1 to 1000
From Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

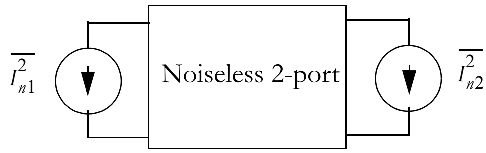
Computational Details

This measurement computes the *current* correlation matrix of an N-port. For a 2-port, for example, this matrix takes the form:

The circuit equivalent of a noisy 2-port represented by its current correlation matrix

$$C = \begin{bmatrix} \overline{I_{n1}^2} & \overline{I_{n1} I_{n2}^*} \\ \overline{I_{n2} I_{n1}^*} & \overline{I_{n2}^2} \end{bmatrix}$$

is illustrated as follows.


$\overline{I_{n1}^2}$
Noiseless 2-port
$\overline{I_{n2}^2}$

Noise Voltage Correlation Matrix: Cv

Summary

Cv computes the N-port noise voltage correlation matrix of a linear (or DC-linearized nonlinear) network. The correlation matrix is normalized to $4kT_0$, where T_0=290K.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
To Port Index	Integer	1 to 1000
From Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value. The complex measurement can be displayed as a real value by specifying the magnitude (**Mag.**), **Angle**, **Real** or imaginary (**Imag.**) component in the Add/Modify Measurement dialog box. The real value can also be displayed in dB by selecting the **dB** checkbox in this dialog box.

Computational Details

This measurement computes the *voltage* correlation matrix of an N-port. For a 2-port, for example, this matrix takes the form:

$$C = (Y^{-1})Ci(Y^{H(-1)})$$

Where Y^H is the Hermetian of Y matrix and *Ci* is Noise Current Correlation matrix. See *Ci* measurement help for more details.

Optimum Noise Figure Match: GMN

Summary

GMN can be used to compute the optimum source reflection coefficient that will provide the minimum noise figure. The measurement is also one of the four coefficients required to define the noise properties of a 2-port. The relation between the noise parameters and the noise figure can be expressed as

$$F = F_{min} + 4 \cdot \frac{R_N}{Z_0} \cdot \frac{|\Gamma_s - \Gamma_{opt}|^2}{|1 + \Gamma_{opt}|^2 (1 - |\Gamma_s|^2)}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Noise Factor: NF

Summary

NF computes the network noise factor as a ratio. To obtain the 'dB' noise figure, select the **DB** check box in the Add/Modify Measurement dialog box. This measurement uses the port termination values for the source impedance when computing the noise factor.

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Minimum Noise Figure: NFMin

Summary

NFMin computes the minimum noise factor as a ratio. To obtain the 'dB' noise figure, select the **DB** check box in the Add/Modify Measurement dialog box. This measurement computes what the minimum noise factor would be with an optimum source termination. The measurement is also one of the three coefficients required to define the noise properties of a 2-port. The relation between the noise parameters and the noise figure is

$$F = F_{min} + \frac{R_N}{G_s}\left|Y_s - Y_{opt}\right|^2$$

where the source termination is given as

$$Y_s = G_s + jB_s$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Noise Measure: NMEAS

Summary

The noise measure of a network represents the noise factor of an infinite cascade of the networks. The noise measure is computed from

$$NMEAS = \frac{NF - 1}{1 - \frac{1}{G_a}}$$

where G_a is the available gain and NF is the noise factor. This measurement uses the port termination values for the source impedance when computing the noise factor. This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Noise Resistance: RN

Summary

RN computes the noise resistance of a two port (un-normalized). This

measurement computes one of the three coefficients required to define the noise properties of a 2-port. The relation between the noise parameters and the noise figure is

$$F = F_{min} + \frac{R_N}{G_s}\left|Y_s - Y_{opt}\right|^2$$

where the source termination is given as

$$Y_s = G_s + jB_s$$

and F_{min} is given as a ratio (not dB).

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Computational Details

From linear noise theory, G_{Sopt}, F_{MIN}, and R_N are not independent of one another. The following condition must be true:

$$RN \geq \frac{Fmin - 1}{4Gsopt}$$

If this equation is not satisfied, RN will be set equation to the right hand side of the equation and a warning will be produced.

Equivalent Input Noise Temperature: TE

Summary

TE returns the equivalent input noise temperature of the 2-port in Kelvin. The input noise temperature is computed from

$$TE = T_0(NF - 1)$$

Where T_0 is standard temperature (290 Kelvin) and *NF* is the noise factor. This measurement uses the port termination values for the source impedance when computing the noise factor.

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Equivalent Output Noise Temperature: TN

Summary

TN returns the equivalent output noise temperature of the 2-port in Kelvin. The output noise temperature is computed from

$$TN = G_a TE$$

Where TE is the equivalent input noise temperature and G_a is the available gain. This measurement uses the port termination values for the source impedance when computing the noise factor.

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in this dialog box.

Optimum Admittance for Noise Match: YMN

Summary

YMN can be used to compute the optimum source admittance that will provide the minimum noise figure. YMN can also be used as one of the three coefficients required to define the noise properties of a 2-port. The relation between the noise parameters and the noise figure is

$$F = F_{min} + \frac{R_N}{G_s}\left|Y_s - Y_{opt}\right|^2$$

where the source termination is given as

$$Y_s = G_s + jB_s$$

and F_{min} is given as a ratio (not dB).

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Optimum Impedance for Noise Match: ZMN

Summary

ZMN can be used to compute the optimum source impedance that will provide the minimum noise figure. ZMN can also be used as one of the three coefficients required to define the noise properties of a 2-port. The relation between the noise parameters and the noise figure is

$$F = F_{min} + \frac{g_N}{R_s}\left|Z_s - Z_{opt}\right|^2$$

where the source termination is given as and

$$Z_s = R_s + jX_s$$

and F_{min} is given as a ratio (not dB).

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Two Port Name	Subcircuit	Two ports only

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Chain Matrix (ABCD-Parameters): ABCD

Summary

The ABCD-parameters represent the chain matrix parameters for a two-port. An example two port ABCD matrix is given as:

$$\begin{bmatrix} v_1 \\ i_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} v_2 \\ -i_2 \end{bmatrix}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only
To Port Index	Integer	1 to 2
From Port Index	Integer	1 to 2

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box.

G-Parameters: G

Summary

The G-parameters are the inverse of the hybrid parameters for a two-port. An example two port G-parameter matrix is given as:

$$\begin{bmatrix} i_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} g_{11} & g_{12} \\ g_{21} & g_{22} \end{bmatrix} \begin{bmatrix} v_1 \\ i_2 \end{bmatrix}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only
To Port Index	Integer	1 to 2
From Port Index	Integer	1 to 2

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **DB** check box.

Hybrid Parameters (H-Parameters): H

Summary

The H-parameters represent the hybrid parameters for a two-port. An example two port H-parameter matrix is given as:

$$\begin{bmatrix} v_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} i_1 \\ v_2 \end{bmatrix}$$

This measurement is applicable to 2-port circuits only.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	Two ports only
To Port Index	Integer	1 to 2
From Port Index	Integer	1 to 2

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Scattering Coefficients (S-Parameters): S

Summary

The S-parameters represent the scattering coefficients for the N-port. A 50 ohm reference impedance is assumed in the computation of the S-parameters unless a port termination other than 50 ohms is specified in the schematic. The termination impedance that can be specified in EMSight is NOT used to normalize the S parameters and all S parameters from EMSight assume a 50 ohm reference impedance.

An example two port S-parameter matrix is given as:

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
To Port Index	Integer	1 to 1000
From Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

S-Parameter Phase Delta: SDeltaP

Summary

SDeltaP is used to compute the difference in phase between two S-parameter values. The phase of the s-parameter specified in the **To port** and **From port** is calculated for both **Data Source Names** and then the difference is calculated. This function will always display the continuos phase difference correctly (if an output equation is used, there may be discontinuities when the phase changes from 180 to -180 degrees for example).

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Data Source Name	Subcircuit	1 to 1000 ports
To port	Integer	1 to 1000
From port	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. in angle units. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Error Between S-Parameters: SModel

Summary

SModel is used to compute the weighted difference between two sets of S-parameters. The weighted difference (error function) calculation method is selected by setting the "Error Function" measurement parameter. This measurement can be used as a goal for optimization when fitting a circuit to measured S-parameter data.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
Data Source Name	Subcircuit	1 to 1000 ports
Error Function	List of options	Average L1 Norm Average L2 Norm Maximum L1 Norm Average Normalized L1 Norm

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Computational Details

The various error functions are calculated as follows:

Average L1 Norm. The weighted difference is the average magnitude of the difference between each element of the S-parameter matrix:

$$Error = \frac{\sum_{i=1}^{N}\sum_{j=1}^{N}(|S_{ijA} - S_{ijB}|)}{N^2}$$

Average L2 Norm. The weighted difference is the average *squared* magnitude of the difference between each element of the S-parameter matrix:

$$Error = \frac{\sum_{i=1}^{N}\sum_{j=1}^{N}(|S_{ijA} - S_{ijB}|)^2}{N^2}$$

Maximum L1 Norm. The maximum difference is the magnitude of the maximum difference between each element of the S-parameter matrix (the magnitude of the largest difference between any pair of entries in the S-parameter matrices):

$$Error = max(|S_{ijA} - S_{ijB}|)$$

Average Normalized L1 Norm. The magnitude of the difference between each element of the S-parameter matrix is calculated. Each difference is then normalized by the average magnitude of the two matrix elements (one from each set):

$$Error = \frac{\sum_{i=1}^{N}\sum_{j=1}^{N}\left(\frac{|S_{ijA} - S_{ijB}|}{0.5 \cdot (|S_{ijA}| + |S_{ijB}|)}\right)}{N^2}$$

In the previous equations, S_A and S_B are the two NxN S-parameter matrices.

Options

The two documents specified by the **Data Source Name** parameters must have the same number of sweep (frequency) points.

Admittance Parameters (Y-Parameters): Y

Summary

The Y-parameters represent the admittance parameters for the N-port. An example two port Y-parameter matrix is given as:

$$\begin{bmatrix} i_1 \\ i_2 \end{bmatrix} = \begin{bmatrix} y_{11} & y_{12} \\ y_{21} & y_{22} \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \end{bmatrix}$$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
To Port Index	Integer	1 to 1000
From Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Impedance Parameters (Z-Parameters): Z

Summary

The Z-parameters represent the impedance parameters for the N-port. An example two port Z-parameter matrix is given as:

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} z_{11} & z_{12} \\ z_{21} & z_{22} \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix}$$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to 1000 ports
To Port Index	Integer	1 to 1000
From Port Index	Integer	1 to 1000

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Time Domain Reflectometry (TDR) Band-Pass Impulse Response: TDR_BPI

Summary

TDR_BPI calculates the time domain, band-pass impulse response of a linear, time-invariant network and is similar to traditional time domain reflectometry (TDR) measurements with the exception that measurement of the zero frequency (DC) component is not required (traditional. TDR has this requirement). This measurement is similar to the TDR, band-pass impulse response measurement available on most network analyzers.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to1000 ports
To Port Index	Integer	1 to 100
From Port Index	Integer	1 to 100
Number of Frequency Points	Integer	2 to 16,384
Time Resolution Factor	Integer	1 to 128
Frequency Domain Window	Integer	1 to 6

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value that represents the time domain, band-pass impulse response of the linear network. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the measurement dialog. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Computational Details

In traditional TDR measurements, the device under test (DUT) is excited with a voltage impulse or step and the time domain response is measured. However, network analyzers measure response in the frequency domain and transform to the time domain mathematically via the Inverse Discrete Fourier Transform (IDFT). This measurement is calculated using this technique.

For a band-pass response, s-parameters are measured over a specified frequency band (f_{start} to f_{stop}) in equal steps. This results in a harmonically-related set of data, but does not include the zero frequency point. Thus, the IDFT will produce a complex time domain waveform. The center frequency, f_c, of the band acts as the zero frequency component for the IDFT. Frequencies from f_{start} to f_c act as the "negative" frequencies, while frequencies from f_c to f_{stop} serve as the "positive" frequencies. Therefore, the total number of points used in calculating the IDFT on network analyzers is simply *N*, which is the number of frequency points specified by the analyzer's user. Comparing this to the low-pass response, it can be seen that the band-pass response suffers from worse time domain resolution.

Reflection vs. Transmission Measurements

Setting the **To Port Index** and **From Port Index** parameters to the same index will result in a reflection measurement. Making them different results in a transmission measurement. Note that for reflection measurements, the time axis will represent *two-way* travel time. The time axis for transmission measurements represent one-way travel time.

Frequency Domain Range and Resolution

For this measurement, the start frequency (f_{start}) is the lowest frequency specified in the **Project Frequencies** list, while the stop frequency (f_{stop})is the highest frequency specified in the list. The frequency step is calculated as:

$$f_{step} = \frac{f_{stop} - f_{start}}{N-1}$$

where *N* is the number of frequency points specified by the user via the **Number of Frequency Points** parameter. The total number of points used in calculating the IDFT for this measurement is simply *N*.

Time Domain Range and Resolution

The frequency step determines the "alias-free" range of the measurement. The alias-free range is the amount of time in which measurements can be made before the response is repeated and is inversely proportional to the frequency step. The alias-free time range, t_r, is:

$$t_r = \frac{1}{f_{step}}$$

If the effective dielectric constant, ε_r, of the DUT medium is known, the actual alias-free physical distance, d_r, can be calculated:

$$d_r = \frac{c \cdot t_r}{\sqrt{\varepsilon_r}}$$

where c is the speed of light.

The time resolution (or time step), t_{step}, of the time domain waveform obtained from the IDFT is dependent on the bandwidth of the measurement. For band-pass measurements, the time resolution is given as:

$$t_{step} = \frac{1}{f_{stop} - f_{start}}$$

For this measurement, the time resolution can be increased by setting the Time Resolution Factor parameter. By setting this number to something other than 1, the measurement will zero-pad in the frequency domain, which increases the bandwidth of the measurement without increasing the total power and thus increases the time resolution. Setting this parameter to 2 will double the resolution, setting it to 3 will triple the resolution, and so on. Setting the parameter to 1 results in no zero-padding.

Windowing

A perfect impulse in the time domain requires an infinite bandwidth in the frequency domain. Truncation of the data between f_{start} and f_{stop} in the frequency domain causes a widening of the pulse in the time domain due to abrupt transitions at the start and stop frequencies. The pulse takes on a $(\sin x)/x$ (sinc) shape, and the sidelobes of the sinc function can mask responses which are small in magnitude. A window can be applied in the frequency domain to give less weight to the spectral components near the band edges.

This measurement provides six options for windowing: **None** (also known as a rectangular window), **Lanczos**, **Bartlett**, **Hanning**, **Hamming**, and **Blackman**. Each window results in a different reduction of sidelobe levels.

Options

The range of frequencies used for this measurement always starts at the lowest frequency in the **Project Frequencies** list and ends at the highest frequency in the list. The frequency step is specified by the user.

Time Domain Reflectometry (TDR) Band-Pass Step Response: TDR_BPS

Summary

TDR_BPS calculates the time domain, band-pass step response of a linear, time-invariant network and is similar to traditional time domain reflectometry (TDR) measurements with the exception that measurement of the zero frequency (DC) component is not required (traditional. TDR has this requirement). This measurement is similar to the TDR, band-pass step response measurement available on most network analyzers.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to1000 ports
To Port Index	Integer	1 to 100
From Port Index	Integer	1 to 100
Number of Frequency Points	Integer	2 to 16,384
Time Resolution Factor	Integer	1 to 128
Frequency Domain Window	Integer	1 to 6

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value that represents the time domain, band-pass step response of the linear network.The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the measurement dialog. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Computational Details

This measurement is similar to the band-pass impulse response measurement (**TDR_BPI**). However, the step response is calculated from the impulse response by integration of the impulse response. Please see the Time Domain Reflectometry TDR_BPI measurement for other computational details.

Options

This range of frequencies used for this measurement always starts at the lowest frequency in the **Project Frequencies** list and ends at the highest frequency in the list. The frequency step is specified by the user.

Time Domain Reflectometry (TDR) Low-Pass Impulse Response: TDR_LPI

Summary

TDR_LPI calculates the time domain, low-pass impulse response of a linear, time-invariant network and is similar to traditional time domain reflectometry (TDR) measurements. This measurement is similar to the TDR, low-pass impulse response measurement available on most network analyzers.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to1000 ports
To Port Index	Integer	1 to 100
From Port Index	Integer	1 to 100
Number of Frequency Points	Integer	2 to 16,384
Time Resolution Factor	Integer	1 to 128
Frequency Domain Window	Integer	1 to 6

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value that represents the time domain, low-pass impulse response of the linear network. Although the imaginary part of the result is zero, a complex result is returned to keep this measurement consistent with the other TDR measurements. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the measurement dialog. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Computational Details

In traditional TDR measurements, the device under test (DUT) is excited with a voltage impulse or step and the time domain response is measured. However, network analyzers measure response in the frequency domain and transform to the time domain mathematically via the Inverse Discrete Fourier Transform (IDFT). This measurement is calculated using this technique.

For a low-pass response, s-parameters are computed from DC to the desired stop frequency in equal steps (since network analyzers cannot perform measurements at DC, the DC value is extrapolated). This results in a harmonically-related set of data that includes the zero frequency point. When this data is mirrored about zero frequency in a complex-conjugate fashion, the IDFT will produce a purely real time domain waveform. Therefore, the total number of points used in calculating the IDFT on network analyzers is $2 \cdot N + 1$, where N is the number of frequency points specified by the user of the analyzer. By essentially doubling the number of frequency points used, the time-domain resolution is also doubled which is one advantage of the low-pass response over the band-pass response.

Reflection vs. Transmission Measurements

Setting the **To Port Index** and **From Port Index** parameters to the same index will result in a reflection measurement. Making them different results in a transmission measurement. Note that for reflection measurements, the time axis will represent *two-way* travel time. The time axis for transmission measurements represent one-way travel time.

Frequency Domain Range and Resolution

For this measurement, the stop frequency (f_{stop})is the highest frequency specified in the **Project Frequencies** list. The frequency step is calculated as:

$$f_{step} = \frac{f_{stop}}{N - 1}$$

where N is the number of frequency points specified by the user via the **Number of Frequency Points** parameter. Note that the number of frequency points includes the zero frequency (DC) point. Therefore, the total number of points used in calculating the IDFT for this measurement is $2 \cdot N - 1$.

Time Domain Range and Resolution

The frequency step determines the “alias-free” range of the measurement. The alias-free range is the amount of time in which measurements can be made before the response is repeated and is inversely proportional to the frequency step. The alias-free time range, t_r, is:

$$t_r = \frac{1}{f_{step}}$$

If the effective dielectric constant, ε_r, of the DUT medium is known, the actual alias-free physical distance, d_r, can be calculated:

$$d_r = \frac{c \cdot t_r}{\sqrt{\varepsilon_r}}$$

where c is the speed of light.

The time resolution (or time step), t_{step}, of the time domain waveform obtained from the IDFT is dependent on the bandwidth of the measurement. For low-pass measurements, the time resolution is given as:

$$t_{step} = \frac{1}{2 \cdot f_{stop}}$$

where the factor of 2 is due to the complex-conjugate mirroring of the data. For this measurement, the time resolution can be increased by setting the Time Resolution Factor parameter. By setting this number to something other than 1, the measurement will zero-pad in the frequency domain, which increases the bandwidth of the measurement without increasing the total power and thus increases the time resolution. Setting this parameter to 2 will double the resolution, setting it to 3 will triple the resolution, etc. Setting the parameter to 1 results in no zero-padding.

Windowing

A perfect impulse in the time domain requires an infinite bandwidth in the frequency domain. Truncation of the data between f_{start} and f_{stop} (for low-pass measurements, $f_{start} = -f_{stop}$)in the frequency domain causes a widening of the pulse in the time domain due to abrupt transitions at the start and stop frequencies. The pulse takes on a $(\sin x)/x$ (sinc) shape, and the sidelobes of the sinc function can mask responses which are small in magnitude. A window can be applied in the frequency domain to give less weight to the spectral components near the band edges.

This measurement provides six options for windowing: **None** (also known as a rectangular window), **Lanczos**, **Bartlett**, **Hanning**, **Hamming**, and **Blackman**. Each window results in a different reduction of sidelobe levels.

Options

The range of frequencies used for this measurement always start at DC and end at the highest frequency specified in the **Project Frequencies** list. The frequency step is specified by the user.

Time Domain Reflectometry (TDR) Low-Pass Step Response: TDR_LPS

Summary

TDR_LPS calculates the time domain, low-pass step response of a linear, time-invariant network and is similar to traditional time domain reflectometry (TDR) measurements. This measurement is similar to the TDR, low-pass step response measurement available on most network analyzers.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	1 to1000 ports
To Port Index	Integer	1 to 100
From Port Index	Integer	1 to 100
Number of Frequency Points	Integer	2 to 16,384
Time Resolution Factor	Integer	1 to 128
Frequency Domain Window	Integer	1 to 6

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value that represents the time domain, low-pass step response of the linear network. Although the imaginary part of the result is zero, a complex result is returned to keep this measurement consistent with the other TDR measurements. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the measurement dialog. The real value can also be displayed in dB by selecting the **dB** check box in the Add/Modify Measurement dialog box.

Computational Details

This measurement is similar to the low-pass impulse response measurement (**TDR_LPI**). However, the step response is calculated from the impulse response by integration of the impulse response. Please see the Time Domain Reflectometry TDR_LPI measurement for other computational details.

Options

This range of frequencies used for this measurement always start at DC and end at the highest frequency specified in the **Project Frequencies** list. The frequency step is specified by the user.

Load Pull Contours, Measured: LPCM

Summary

LPCM plots contours on a Smith chart for a selected column of data from a measured Focus (*.lpd) or Maury (*.lp* or *.sp*) load pull data file.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Focus (*.lpd) or Maury (*.lp* or *.sp*) data file
Contour Max	Real	-500 to 500
Contour Min	Real	-500 to 500
Contour Step	Real	0.01 to 100
Load Pull Data Col. Index	Integer	1 to 100
Zo, real	Real	0.1 to 2500 ohms
Zo, imag	Real	-2500 to 2500 ohms

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the parameters of a contour or a set of contours.

Computational Details

The column index does not include the first three columns of a Focus data file or the first two columns of a Maury data file. Therefore, an index of 1 corresponds to the first measured parameter data column.

The complex Zo is assumed to be of the form $Zo = Ro + jXo$. The "Zo, imag" parameter can be negative or positive.

Maximum of Load Pull Contours (Measured): LPCMMAX

Summary

LPCMMAX plots the maximum contour value on a Smith chart for a selected column of data from a measured Focus (*.lpd) or Maury (*.lp* or *.sp*) load pull data file.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Focus (*.lpd) or Maury (*.lp* or *.sp*) data file
Load Pull Data Col. Index	Integer	1 to 100
Zo, real (ohms)	Real	0.1 to 2500 ohms
Zo, imag (ohms)	Real	-2500 to 2500 ohms

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the maximum value of a set of contours.

Computational Details

The column index does not include the first three columns of a Focus data file or the first two columns of a Maury data file. Therefore, an index of 1 corresponds to the first measured parameter data column.

The complex Zo is assumed to be of the form $Zo = Ro + jXo$. The "Zo, imag" parameter can be negative or positive.

Minimum of Load Pull Contours (Measured): LPCMMIN

Summary

LPCMMIN plots the minimum contour value on a Smith chart for a selected column of data from a measured Focus (*.lpd) or Maury (*.lp* or *.sp*) load pull data file.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Focus (*.lpd) or Maury (*.lp* or *.sp*) data file
Load Pull Data Col. Index	Integer	1 to 100
Zo, real (ohms)	Real	0.1 to 2500 ohms
Zo, imag (ohms)	Real	-2500 to 2500 ohms

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the minimum value of a set of contours.

Computational Details

The column index does not include the first three columns of a Focus data file or the first two columns of a Maury data file. Therefore, an index of 1 corresponds to the first measured parameter data column.

The complex Zo is assumed to be of the form $Zo = Ro + jXo$. The "Zo, imag" parameter can be negative or positive.

Load Pull Contours, Simulated: LPCS

Summary

LPCS plots contours on a Smith chart for a tab-delimited, text data file. This measurement is generally only used with the AWR **Load Pull Wizard**.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Tab-delimited, text data file
Contour Max	Real	-500 to 500
Contour Min	Real	-500 to 500
Contour Step	Real	0.01 to 100

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the parameters of a contour or a set of contours.

Computational Details

The tab-delimited, text data file contains an m x n matrix of data which represents a grid of points in the real-imaginary plane from -1 to +1 along each axis.

Maximum of Load Pull Contours (Simulated): LPCSMAX

Summary

LPCSMAX plots the maximum contour value on a Smith chart for a tab-delimited, text data file. This measurement is generally only used with the AWR **Load Pull Wizard**.

Parameters

Name	Type	Range
Data File Name	Subcircuit	Tab-delimited, text data file

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the maximum value of a set of contours.

Computational Details

The tab-delimited, text data file contains an *m* x *n* matrix of data which represents a grid of points in the real-imaginary plane from -1 to +1 along each axis. The contours are computed and the maximum value is then found.

Minimum of Load Pull Contours (Simulated): LPCSMIN

Summary

LPCSMIN plots the minimum contour value on a Smith chart for a tab-delimited, text data file. This measurement is generally only used with the AWR **Load Pull Wizard**.

Parameters

Name	Type	Range
Data File Name	Subcircuit	Tab-delimited, text data file

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns the minimum value of a set of contours.

Computational Details

The tab-delimited, text data file contains an m x n matrix of data which represents a grid of points in the real-imaginary plane from -1 to +1 along each axis. The contours are computed and the maximum value is then found.

Load Pull Gamma Points, Measured File: LPGPM

Summary

LPGPM plots the reflection coefficient points (impedance points) from a measured Focus (*.lpd) or Maury (*.lp* or *.sp*) load pull data file.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Focus (*.lpd) or Maury (*.lp* or *.sp*) data file
Zo, real	Real	0.1 to 2500 ohms
Zo, imag	Real	-2500 to 2500 ohms

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns a complex value.

Computational Details

The data file must be in standard Focus or Maury format. The data must be in either complex impedance or complex reflection coefficient format. For Focus format, the real part is listed in column two and the imaginary part in column three (column one is ignored). For Maury format, the real part is listed in column one and the imaginary part in column two.

If the original data is in reflection coefficient form, Z is first calculated as:

$$Z = Zo_{df} \cdot \left(\frac{1+\Gamma_{df}}{1-\Gamma_{df}}\right)$$

Where Zo_{df} and Γ_{df} are the system impedance and reflection coefficients, respectively, of the data file. The final reflection coefficient data that displays is then normalized to the specified complex Zo: $\Gamma = \frac{Z - Zo^*}{Z + Zo}$.

If the original data is in impedance format, this same equation is used to convert the data to reflection coefficient format.

The complex Zo is assumed to be of the form $Zo = Ro + jXo$. The "Zo, imag" parameter can be negative or positive.

Load Pull Gamma Points, 2-Column Tabular File: LPGPT

Summary

LPGPT plots the reflection coefficient points (impedance points) from a two-column, tab-delimited, text data file. The first column specifies the real part of the reflection coefficient, and the second column specifies the imaginary part.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Any two-column, tabular data file
Zo File, re	Real	0.1 to 2500 ohms
Zo File, im	Real	-2500 to 2500 ohms
Zo Disp, re	Real	0.1 to 2500 ohms
Zo Disp, im	Real	-2500 to 2500 ohms

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

Result

This measurement returns a complex value.

Computational Details

The data in the data file must be in complex reflection coefficient format, with the real part in the first column and the imaginary part in the second column. You can use the "!" character at the beginning of a line to specify it as a comment line. All comment lines are ignored.

The reflection coefficients are normalized to the specified Zo. First, Z is calculated as:

$$Z = Zo_{df} \cdot \left(\frac{1 + \Gamma_{df}}{1 - \Gamma_{df}}\right)$$

Where Zo_{df} and Γ_{df} are the complex system impedance and reflection coefficients, respectively, of the data file. The reflection coefficient data that is displayed is then normalized to the specified complex Zo for display: $\Gamma = \frac{Z - Zo^*}{Z + Zo}$.

The complex system impedances are assumed to be of the form $Z = R + jX$. The Zo File, im and Zo Disp, im parameters can be negative or positive.

Interpolated Load Pull Data: LPINT

Summary

LPINT determines the interpolated value of a selected measured parameter (output power, PAE, etc.). The user specifies the measured load (or source) pull data file (representing the device) and a matching circuit document (schematic, Touchstone data file, or EM structure). The matching circuit document determines the impedance seen by the device.

Parameters

Name	Type	Range
Load Pull Data File Name	Subcircuit	Focus (*.lpd) or Maury (*.lp* or *.sp*) data file
Data Source Name	Subcircuit	2 to 1000 ports
Output Port Number	Integer	1 to 1000
Load Pull Data Col. Index	Integer	1 to 100

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Load Pull measurements are based off of data files and so swept parameters are not defined.

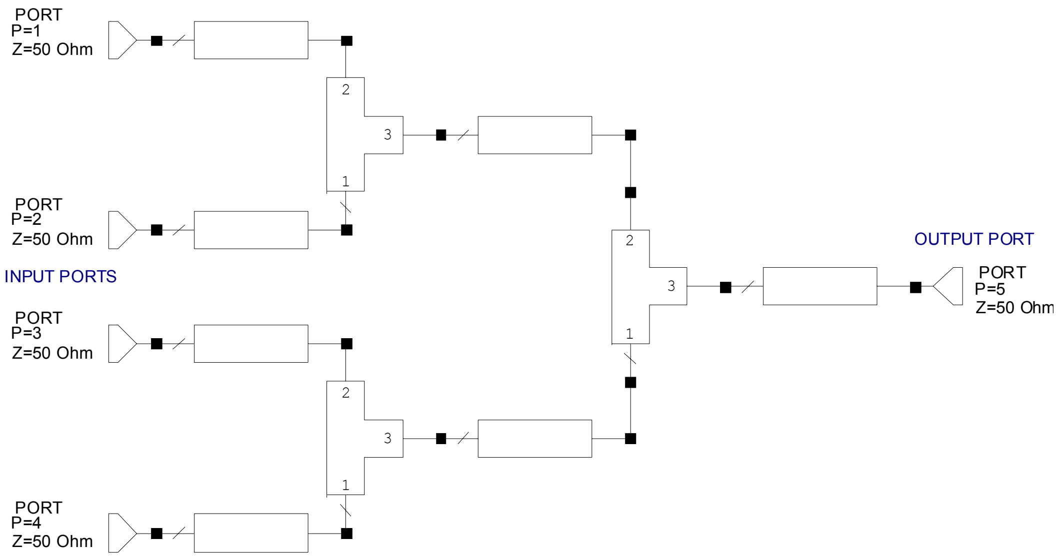
Result

This measurement returns a real value.

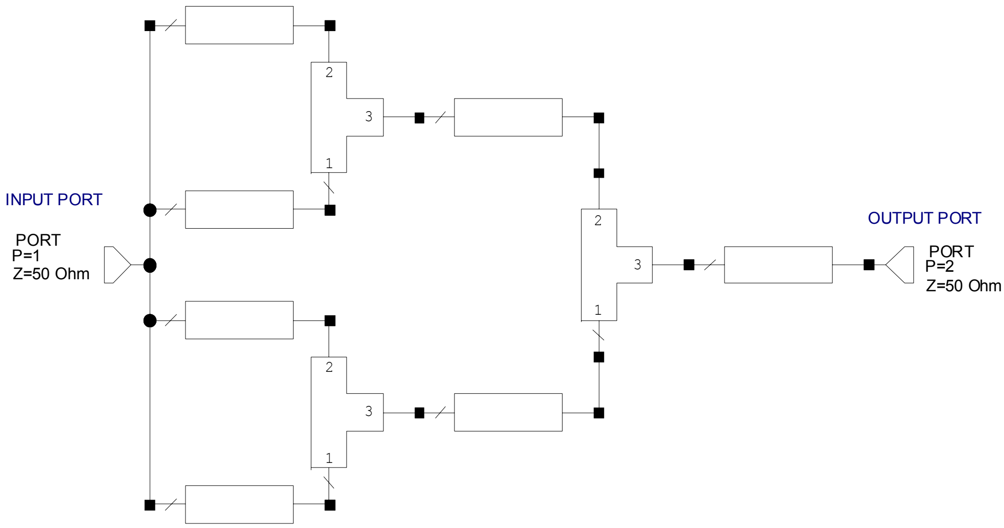
Computational Details

This measurement computes the complex impedance seen looking into an "input" port on a schematic. Each input port represents a connection to a measured device. Since impedance is the independent variable of a load (or source) pull measurement, this impedance is then used to calculate the interpolated value from within a column of data in the data file. If the schematic contains N ports (including the output port), an N-port to 2-port conversion is performed. This allows each branch of a circuit to be optimized without being loaded by the other branches. This is useful when connecting multiple devices via power combiners/dividers.

The following is an example circuit with four branches:



The measurement converts the previous circuit to the following equivalent 2-port circuit:



The input impedance looking into an input port is the calculated input impedance of the resulting 2-port divided by the number of input ports (N-1), or:

$$Zin = \frac{Zin'}{N-1}$$

Where N is the total number of ports in the schematic.

The reflection coefficient is then computed as follows:

$$\Gamma = \frac{Zin - Zo_{port}}{Zin + Zo_{port}}$$

where Zo_{port} is the termination impedance of an input port. Interpolation is then performed at this reflection coefficient in the interpolation algorithm.

This measurement uses the “thin plate spline” two-dimensional interpolation algorithm with no smoothing.

IMPORTANT ASSUMPTIONS:

1. This measurement allows the user to specify only one “output” port. All other ports are considered “input” ports.
2. It is assumed that all input ports have the same termination impedance, Zo.
3. It is assumed that all branches are identical to one another.

DC Linearized Capacitance: CDC

Summary

CDC computes the capacitances of a nonlinear branch at the DC operating point.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Controlling Branch	Integer	1-10

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in charge units.

Computational Details

Nonlinear elements are described by their charge-voltage and current-voltage relations of the form

$$q(v_1, v_{2, \ldots}) \qquad i(v_1, v_2, \ldots)$$

Capacitances are partial derivatives of charge-voltage expressions and are given by

$$c_1(v_1, v_{2, \ldots}) = \frac{\partial}{\partial v_1} q(v_1, v_{2, \ldots}) \qquad etc \quad .$$

The "controlling branch" parameter identifies the controlling voltage with respect to which the differentiation is performed. In more complicated cases the end-user is usually unaware of the order of the controlling branches in the Microwave Office/Analog Office implementation of a particular charge function, so this measurement finds its primary use in internal development of nonlinear devices. There are several cases where this measurement has practical use, however; for example, it can be used for straightforward fitting of varactor diode characteristics using optimization.

Notes:

- For the measurement to work, you must select the **Q, C, and G for nonlinear srcs** check box in the Advanced Harmonic Balance Options dialog box. To access this dialog box, choose **Options > Default Circuit Options** to display the Circuit Options dialog box. Click the **Harmonic Balance** tab and then click the **Advanced** button to display the Advanced Harmonic Balance Options dialog box.

- The measurement invokes a DC simulation on the selected data source and returns the corresponding DC capacitance. This quantity is generally different than the DC component of the capacitance waveform under large signal conditions.

DC Charge: QDC

Summary

QDC computes the charge stored in a nonlinear branch at the DC operating point.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in charge units.

Computational Details

Nonlinear elements are described by their charge-voltage and current-voltage relations of the form

$$q(v_1, v_2, \ldots) \qquad i(v_1, v_2, \ldots)$$

This measurement displays $q(v_1, v_2, \ldots)$ for a selected nonlinear branch.

NOTE.

- For this measurement to work, you must select the **Q, C, and G for nonlinear srcs** check box in the Advanced Harmonic Balance Options dialog box. To access this dialog box, choose **Options > Default Circuit Options** to display the Circuit Options dialog box. Click the **Harmonic Balance** tab and then click the **Advanced** button to display the Advanced Harmonic Balance Options dialog box.

- This measurement invokes a DC simulation on the selected data source and returns the corresponding DC charge. This quantity is generally different than the DC component of the charge waveform under large signal conditions.

DC Linearized Conductance: GDC

Summary

GDC computes the conductances of a nonlinear branch at the DC operating point.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Controlling Branch	Integer	1-10

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in conductance units.

Computational Details

Nonlinear elements are described by their charge-voltage and current-voltage relations of the form

$$q(v_1, v_{2, \ldots}) \qquad i(v_1, v_2, \ldots)$$

Conductances are partial derivatives of current-voltage expressions and are given by

$$g_1(v_1, v_{2, \ldots}) = \frac{\partial}{\partial v_1} i(v_1, v_{2, \ldots}) \qquad etc.$$

The "controlling branch" parameter identifies the controlling voltage with respect to which the differentiation is performed. In more complicated cases the end-user is usually unaware of the order of the controlling branches in the Microwave Office/ Analog Office implementation of a particular current function, so this measurement finds its primary use in internal development of nonlinear devices.

NOTE.

- For the measurement to work, you must select the **Q, C, and G for nonlinear srcs** check box in the Advanced Harmonic Balance Options dialog box. To access this dialog box, choose **Options > Default Circuit Options** to display the Circuit Options dialog box. Click the **Harmonic Balance** tab and then click the **Advanced** button to display the Advanced Harmonic Balance Options dialog box.

- The measurement invokes a DC simulation on the selected data source and returns the corresponding DC capacitance. This quantity is generally different than the DC component of the capacitance waveform under large signal conditions.

Current Harmonic Component: Icomp

Summary

Icomp is used to measure a harmonic component of the current measured at a specified point in the circuit specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. The current value is returned as the complex magnitude of the current component at the harmonic frequency (to obtain the RMS. value, you must divide by the square root of 2).

To obtain the DC current, use a harmonic index of zero (this will be the DC value with AC sources present, so it will capture self biasing). To obtain the current at the fundamental frequency use a harmonic index of one.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may display in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in current units. 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 real value can also be displayed in dB by selecting the **dB** check box.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

DC Current: IDC

Summary

IDC measures the DC current at a specified point in the circuit specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated.

This DC value is a result of DC simulation only without the presence of any AC signals. Therefore, it will not show any self biasing effects. If you want to see self biasing, please use the Icomp measurement with a harmonic index of 0.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units.

Current Envelope: Ienv

Summary

Ienv displays the complex envelope of a current waveform specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. The measurement is analogous to Nonlinear Voltage\Venv. Please consult Venv documentation for additional details

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
No. Samples	Integer value	N/A
No. Periods	Integer value	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 x-axis for this measurement is always in time units.

Current Eye Diagram: Ieye

Summary

Ieye displays the time domain current measured at a specified point in the circuit specified by the **Measurement Component** parameter in the form of an eye diagram. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. This measurement is applicable only to circuits that are driven by PORT_ARBS, PORT_PRBS and PORT_SIG elements. Please refer to PORT_SIG for signal file modifications needed to accommodate eye diagram displays.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units. The x-axis for this measurement is always in time units.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

Differential Current Eye Diagram: IeyeD

Summary

IeyeD displays the time domain current measured at a specified point in the circuit specified by the +Measurement Component and -Measurement Component parameters in the form of an eye diagram. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. This measurement is applicable only to circuits that are driven by PORT_ARBS, PORT_PRBS and PORTSIG elements. Please see PORTSIG for signal file modifications needed to accommodate eye diagram displays.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units. The x-axis for this measurement is always in time units.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

FFt of Current for Specified Period: Ifft

Summary

Ifft calculates the spectrum of the specified periodic current waveform using FFT. It is intended to be used with transient simulators, such as HSPICE transient or Spectre transient.

You need to specify the Start and End Time, thus the fundamental period T is determined as T=End Time-Start Time, and the fundamental frequency f_0=1/T. The Start and End Time options allow skipping of the transient processes, so that FFT is applied to the portion of the transient waveform that corresponds to the steady state. The Start and End Time are specified in time units explicitly. Seconds (s), milliseconds (ms), microseconds (us), nanoseconds (ns), or picoseconds (ps) can be specified as units for Start and End Time.

The number of harmonics N specifies the highest frequency (Nf_0) in the calculated spectrum.

Since the FFT requires evenly spaced time samples while transient waveforms generated by HSPICE or Spectre have variable time step, interpolation of transient waveforms is performed. Two interpolation methods (Spline or Linear) can be selected. The default interpolation method (Spline) is recommended in most cases while Linear interpolation is used if the number of available time points in the transient waveform is extremely small. The latter condition occurs infrequently.

The oversampling factor improves accuracy of the calculated spectrum and avoids aliasing. Thus the minimum allowed oversampling factor is 2 to avoid aliasing. The default value (4) is adequate in most cases.

Classic time domain windows can be applied to the waveform prior to calculating the FFT. The default is no windowing (this is optimal if the fundamental frequency is known).

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port, current probe, or a terminal of a circuit element	N/A

Name	Type	Range
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Number of harmonics	Integer number	N>1. The highest frequency in the spectrum is N/(End Time - Start Time)
Interpolation method	String	Spline (recommend in most cases), or Linear
Oversampling factor	Integer Number	2 to 16
Time Domain Window	String	Multiple classic time domain windows are available.

Result

This measurement returns N+1 complex numbers (approximate Fourier components of the specified current). The notation is

$$i(t) = I_0 + Re\left[\sum_{k=1}^{N} I_k \exp[jk\omega_0 t]\right]$$

where N is the number of harmonics, and I_k (k=0, …, N) are the complex Fourier components. If dB display is selected, $20\log_{10}|I_k|$ is displayed.

Graph Type

This measurement can be displayed on a rectangular graph or table. Document frequency is one of the possible sweep variables.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -

90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Frequency Domain Current: Iharm

Summary

Iharm is used to measure the current spectrum specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. The current value is returned as a spectrum of complex magnitudes of all the current components at each harmonic frequency (to obtain the RMS. value, you must divide by the square root of 2).

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in current units. 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 real 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 always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Time Domain Current: Itime

Summary

Itime is used to measure the time domain current specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the current measured will be the current entering this node. If the measurement component parameter specifies just an element, an error will be generated. The current value is returned as a real valued time waveform of the current values. For one-tone analysis, two periods of the waveform will be displayed.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Offset	List of options	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. in current units. The x-axis for this measurement is always in time units.

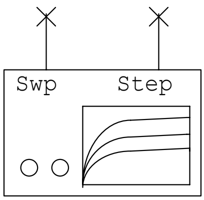
Computational Details

The **Offset** setting controls any desired shift in the waveform. “None” will introduce no shift. “First Point” will offset by the first waveform point value so the first point will always be 0. “Average” will offset by the average value of the waveform. “RMS” will offset by the Root Mean Square (RMS) value of the waveform.

I-V Curve Trace I at Swept Terminal: IVCurve

Summary

The I-V curve trace measurement is a specialized measurement that requires the presence of a single IVCURVE measurement element in the schematic. The IVCURVE measurement element is used just like a real curve tracer would be used. The IVCURVE measurement element has two terminals, one for a swept voltage, and the other for a stepped voltage. The following is the schematic symbol for the IVCURVE element.



This measurement is controlled by the settings associated with the IVCURVE measurement element. For more information on the settings, see the documentation for the IVCURVE element.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

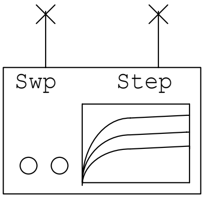
Result

This measurement returns a DC-IV curve trace..

I-V Curve Trace I at Stepped Terminal: IVCurve2

Summary

The I-V curve trace measurement is a specialized measurement that requires the presence of a single IVCURVE measurement element in the schematic. The IVCURVE measurement element is used just like a real curve tracer would be used. The IVCURVE measurement element has two terminals, one for a swept voltage, and the other for a stepped voltage. The following is the schematic symbol for the IVCURVE element.



This measurement is controlled by the settings associated with the IVCurve measurement element. This measurement differs from the IVCurve measurement in that it displays the current from the stepped terminal. For more information on the settings, see the IVCurve element documentation.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

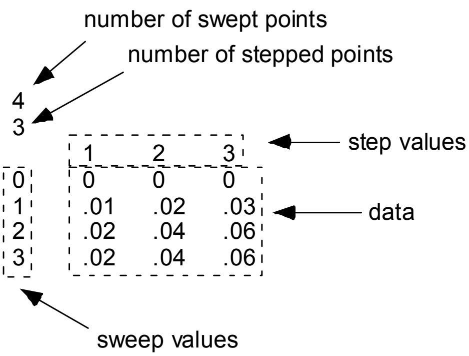
This measurement returns a DC-IV curve trace.

I-V Curve Delta I at Swept Terminal: IVDelta

Summary

IVDelta computes the difference between measured and model-predicted results of I-V measurements; it is useful for nonlinear model parameter extraction.

The measured results are stored in a data file which is imported by right-clicking **Data Files** in the Project Browser. The file name extension should be .ivd. The following example shows the format of the data file.



The measurement has two parameters indicating the sources of IV data. Normally, one source is the measurement data file and the other is the schematic which contains the simulated results obtained using the IVCurve (or IVCurveI) meter.

IVDelta computes, at each point in the sweep, the sum of absolute values of the difference between the measured and modeled results.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

I-V Curve Delta I at Stepped Terminal: IVDelta2

Summary

IVDelta2 is analogous to IVDelta; the difference is that current, not voltage, is used as the stepped variable.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

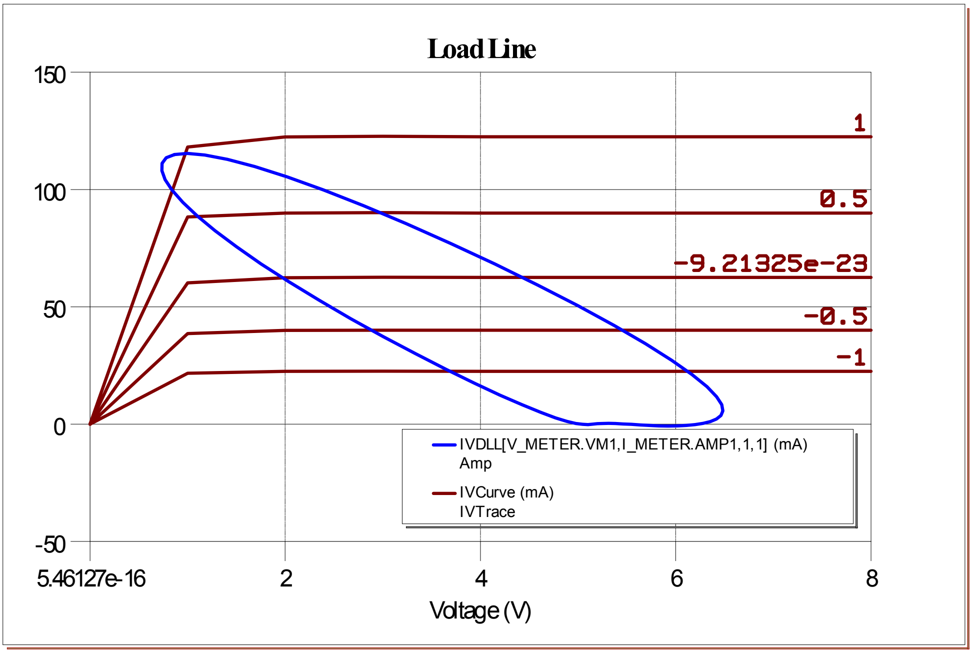
Result

This measurement returns a real value.

I-V Dynamic Load Line: IVDLL

Summary

The dynamic load line measurement is used to plot the dynamic I-V trajectory on a rectangular graph. Typically, the load line measurement is used in conjunction with the IVCurve measurement as shown below.



The DC IV curves will be generated from a simple schematic with an IVCURVE element (or swept variables) and an active device. The dynamic load line measurement will be specified by the **Voltage Measurement Component** and **Current Measurement Component** parameters using the circuit of the entire design (e.g., a power amplifier circuit) that uses that same device. By putting both the IVCURVE measurement and the IVDLL measurement on the same graph, the relationship between the static and dynamic IV characteristics can be displayed.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Voltage Measure Component	String	N/A
Current Measure Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in current units.

DC Linearized Resistance: RDC

Summary

RDC computes the resistances of a nonlinear branch at the DC operating point.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	0 to 1000
Measurement Component	String	N/A
Controlling Branch	Integer	1-10

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in resistance units.

Computational Details

Nonlinear elements are described by their charge-voltage and current-voltage relations of the form

$$q(v_1, v_2, \ldots) \qquad i(v_1, v_2, \ldots)$$

Small signal resistances are given by

$$r_1(v_1, v_2, \ldots) = \frac{1}{\frac{\partial}{\partial v_1} i(v_1, v_2, \ldots)} \qquad etc.$$

The "controlling branch" parameter identifies the controlling voltage with respect to which the differentiation is performed. In more complicated cases the end-user is usually unaware of the order of the controlling branches in the Microwave Office/Analog Office implementation of a particular current function, so this measurement finds its primary use in internal development of nonlinear devices.

NOTE.

- For the measurement to work, you must select the **Q, C, and G for nonlinear srcs** check box in the Advanced Harmonic Balance Options dialog box. To access this dialog box, choose **Options > Default Circuit Options** to display the Circuit Options dialog box. Click the **Harmonic Balance** tab and then click the **Advanced** button to display the Advanced Harmonic Balance Options dialog box.

- The measurement invokes a DC simulation on the selected data source and returns the corresponding DC capacitance. This quantity is generally different than the DC component of the capacitance waveform under large signal conditions.

Conversion Gain: ConvG

Summary

ConvG computes the small-signal conversion gain of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer value	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower
Noise Frequency Index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value.

Computational Details

Please see ConvG_F documentation for details on this measurement. ConvG_F displays the results as a function of the noise frequency, as opposed to ConvG which is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

Conversion Gain (Sweep Over Noise Frequency): ConvG_F

Summary

ConvG_F computes the small-signal conversion gain of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

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

Computational Details

Conversion gain is computed from the port defined by the PortFrom parameter of the NLNOISE model to the port defined by the PortTo parameter. The output signal frequency is defined by Output large-signal harmonic/Output sideband parameters. The input signal frequency is defined by Input large-signal harmonic/Input sideband parameters.

This conversion gain computation is used, e.g., in mixers in which the power of the input signal is appreciably smaller than that of the LO. The ratio of the LO power and small signal power should be at least 10 dB, preferably 15 dB or more. If this is not the case the conversion gain should be computed with the LSSnm measurement.

Nonlinear Noise Figure: NF_SSB0

Summary

NF_SSB0 computes a large-signal noise figure of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower
Noise Frequency index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

Please see NF_SSB0_F documentation for details on this measurement. NF_SSB0_F displays the results as a function of the noise frequency, as opposed to NF_SSB0 which is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

Noise Figure (Swept Over Noise Frequency): NF_SSB0_F

Summary

NF_SSB0_F computes a large-signal noise figure of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The x-axis frequencies are defined by the NFstart and NFend parameters of the NLNOISE element.

Computational Details

The NF_SSB0_F measurement computes the noise figure of a nonlinear circuit as:

$$NF = \frac{N(f_{out})}{N_s(f_{in} \rightarrow f_{out})}$$

where

- $N(f_{out})$ is the total noise power at the output in 1 Hz bandwidth around frequency f_{out}.

- $N_s(f_{in} \rightarrow f_{out})$ is the noise power at the output in 1 Hz bandwidth around frequency f_{out} due to the input noise source at frequency f_{in}. By convention, the input noise source is a termination resistor R=Rsource at T_0=290 K.

In the case of a down-converting mixer, for example, f_{in} is typically the upper-side-band RF frequency and f_{out} is the IF frequency.

The computation of the output noise power (the numerator in the NF equation) excludes the contribution of the load termination.

The NF_SSB0_F measurement computes the total noise power at the output as

$$N(f_{out}) = kT_0 G_{f_{in} \rightarrow f_{out}} + N_n(f_{out}) \qquad .$$

where $G_{f_{in} \rightarrow f_{out}}$ is the transducer conversion gain from the input port to the output port between f_{in} and f_{out}; and $N_n(f_{out})$ is the noise contributed by the network under test. Note that $N(f_{out})$ does not include the reference source contribution from frequencies other than f_{in} (image frequencies), in disagreement with the IEEE definition of noise figure but in analogy to the linear noise figure definition. For a more general noise figure definition see NF_SSBN_F.

Noise Figure (IEEE Definition): NF_SSBN

Summary

NF_SSBN computes a large-signal noise figure of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower
Noise Frequency index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

See the NF_SSBN_F documentation for details on this measurement. NF_SSBN_F displays the results as a function of the noise frequency as opposed to NF_SSB, which is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

Noise Figure (IEEE Definition) (Swept Over Noise Frequency): NF_SSBN_F

Summary

NF_SSBN_F computes a large-signal noise figure of a nonlinear circuit.

Parameters.

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

The NF_SSBN_F measurement computes the noise figure of a nonlinear circuit as:

$$NF = \frac{N(f_{out})}{N_s(f_{in} \rightarrow f_{out})}$$

where

- $N(f_{out})$ is the total noise power at the output in 1 Hz bandwidth around frequency f_{out}.
- $N_s(f_{in} \rightarrow f_{out})$ is the noise power at the output in 1 Hz bandwidth around frequency f_{out} due to the input noise source at frequency f_{in}. By convention, the input noise source is a termination resistor R=Rsource at T=290 K.

In the case of a down-converting mixer, for example, f_{in} is typically the upper-side-band RF frequency, while f_{out} is the IF frequency.

The computation of the output noise power (the numerator in the NF equation) excludes the contribution of the load termination.

The NT_SSB_F measurement computes the total noise power at the output as

$$N(f_{out}) = kT_0 \sum_f G_{f \rightarrow f_{out}} + N_n(f_{out})$$

where $\sum_f G_{f \rightarrow f_{out}}$ is the sum of transducer conversion gains from the input port to the output port from all image frequencies to f_{out}; and $N_n(f_{out})$ is the noise contributed by the network under test.

Noise Spectrum Density at "Port To" Port: NPo_NL

Summary

NPo_NL computes the spectral density of noise power delivered to the output terminal of a nonlinear circuit, following a nonlinear noise analysis. The output terminal is defined by the PortTo parameter of the NLNOISE element.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

NPo_NL returns a real value in power units per Hz. The measurement can be displayed in dBm/Hz by selecting **DBm** as the **Result Type** in the Add/Modify Measurement dialog box. The x-axis for this measurement is in frequency units. The x-axis frequencies are defined by the NFstart and NFend parameters of the NLNOISE element

Noise Temperature: NT_SSB

Summary

NT_SSB computes the noise temperature of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower
Noise Frequency index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

NT_SSB returns a real value in temperature units.

Computational Details

Please see NT_SSB_F documentation for details on this measurement. NT_SSB_F displays the results as a function of the noise frequency, as opposed to NT_SSB which is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

Noise Temperature (Swept Over Noise Frequency): NT_SSB_F

Summary

NT_SSB_F computes a large-signal noise figure of a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Output sideband	List of options	Upper/Lower
Input large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Input sideband	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

NT_SSB_F returns a real value in temperature units.

Computational Details

The noise temperature is given by

$$T = T_0(F-1)$$

where $T_0 = 290\ \mathrm{K}$ is the input reference temperature and F is the noise figure as computed by NF_SSB_F. See the *NF_SSB_F* documentation for details of noise figure computation.

RMS Noise Voltage in V/sqrt(Hz): NV

Summary

NV computes the rms noise voltage at a node (or across a pair of nodes) in a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for Noise Meas.	String	N/A
Large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband	List of options	Upper/Lower
Noise Frequency Index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in units of V/sqrt(Hz).

Computational Details

See the NV_F documentation for details on this measurement. NV_F displays the results as a function of the noise frequency, as opposed to NV which is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

RMS Noise Voltage in V/sqrt(Hz) (Swept Over Noise Frequency): NV_F

Summary

NV_F computes the RMS noise voltage at a node (or across a pair of nodes) in a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for Noise Meas.	String	N/A
Large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in units of V/sqrt(Hz).

Computational Details

The noise meter element (V_NSMTR) must be connected to the node (or a pair of nodes) where noise is to be measured. The noise meter is located in the **MeasDevice** category of the Element Browser.

Noise Voltage Correlation in V^2/Hz: NVCorr

Summary

NVCorr computes an element of the voltage correlation matrix at a node (or across a pair of nodes) in a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for Noise Meas.	String	N/A
Large-signal harmonic 1[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband 1	List of options	Upper/Lower
Large-signal harmonic 2[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband 2	List of options	Upper/Lower
Noise Frequency Index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in NLNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

See the *NVCorr_F* documentation for details on this measurement. NVCorr_F displays the results as a function of the noise frequency, while NVCorr is displayed against arbitrary variable sweeps. Otherwise, the two measurements are computationally identical.

Noise Voltage Correlation in V^2/Hz (Swept Over Noise Frequency): NVCorr_F

Summary

NVCorr_F computes an element of the voltage correlation matrix at a node (or across a pair of nodes) in a nonlinear circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for Noise Meas.	String	N/A
Large-signal harmonic 1[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband 1	List of options	Upper/Lower
Large-signal harmonic 2[a]	Integer	-Max Harmonics to +Max Harmonics
Sideband 2	List of options	Upper/Lower

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

For the purposes of spot noise simulations, noise waveforms (voltages and currents) are represented as

$$n(t) = \sum_{k=-K}^{K} N_k \exp[j(k\omega_0 + \omega)t],$$

where ω is the noise offset frequency and N_k are pseudo-sinusoidal components of the noisy waveform; thus $\langle |N_k|^2 \rangle$ represents the mean-square value of the noise waveform lying in a 1 Hz bandwidth around $k\omega_0 + \omega$.

The noise voltage correlation matrix is of the form:

$$\begin{bmatrix} C_{\text{-K,-K}} & C_{\text{-K,-K+1}} & \cdots & C_{\text{-K,K}} \\ C_{\text{-K+1,-K}} & C_{\text{-K+1,-K+1}} & \cdots & C_{\text{-K+1,K}} \\ \cdots & \cdots & \cdots & \cdots \\ C_{\text{K,-K}} & C_{\text{K,-K+1}} & \cdots & C_{\text{K,K}} \end{bmatrix}$$

where

$$C_{i,j} = \langle V_i V_j^* \rangle .$$

Note that $\sqrt{C_{\text{k,k}}}$ represents the per-Hertz RMS noise voltage at $k\omega_0 + \omega$ as computed by the NV_F measurement.

The noise meter element (V_NSMTR) must be connected to the node (or a pair of nodes) where noise is to be measured. The noise meter is located in the **MeasDevice** category of the Element Browser.

Phase Noise (Swept Over Noise Frequency): PH_NOISE_NL_F

Summary

PH_NOISE_NL_F computes the phase noise at a port of a driven nonlinear circuit. The port number is the PortTo parameter of the NLNOISE control element that must be placed in the schematic. This measurement can be used in conjunction with the OSC_W_PH_NOISE model to simulate the phase noise out of a circuit (amplifier, mixer, etc.) with phase noise at the input of the circuit. This allows oscillation simulations to be run on a smaller scale while still carrying the phase noise through the rest of the circuitry driven by the oscillator.

Parameters

Name	Type	Range
Data Source Name	String	N/A.
Output large-signal harmonic[a]	Integer	-Max Harmonics to +Max Harmonics.

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a positive real value.

The following are the values of frequency:

- For one-tone analysis, $kf_o + \Delta f$, where k is the index specified in Output large signal harmonic, and f_o is the fundamental frequency of the schematic (document).
- For two-tone analysis, $kf_o+mf_1+\Delta f$, where (k, m) are the pair of indices specified under Output large signal harmonic, f_o is the Tone 1 fundamental frequency of the schematic (document), f_1 is the Tone 2 frequency, and Δf is the offset frequency specified in NLNOISE element.

The offset frequencies are swept as specified in the NLNOISE element, thus forming a function L(f) that can be plotted or tabulated.

Computational Details

This measurement implements calculations of the phase noise $L(\omega)$ according to the expression

$$L(\omega) = \frac{V_{k,k} + V_{-(k),k} - 2Re[V_{(-k),k}\exp[2\varphi j]]}{|V_{o,k}|^2}$$

where $V_{0,k}$ is the large signal noiseless voltage corresponding to the harmonic k at the output port, $\varphi = \mathrm{Arg}[V_{0,k}]$, $V_{k,k}$ is the upper sideband of the noise voltage corresponding to the harmonic k, and $V_{-k,-k}$ is the lower sideband of the noise voltage corresponding to the harmonic k.

This calculation is based on the conversion matrix approach as described in V. Rizzoli, A. Costanzo, D. Massoti, F. Mastri, *Computer-Aided Analysis of Near-Carrier Noise in RF-Microwave Frequency Converters*, and in. J RF and Microwave CAE, V. 9, p. 449-467, 1999.

DC Operating Point: OP_DC

Summary

The OP_DC measurement can be used to measure the available DC operating point quantities. This measurement computes the operating point at the DC bias level of the circuit. A typical operating point quantity would be the gm of a transistor for example. The available operating point quantities can be selected in the **Measurement Component** drop-down list. Additional operating point quantities can be selected from the browser by clicking the **...** button next to the **Measurement Component** drop-down.

NOTE: The **Operating point info** check box in **Compute and save results from** must be selected in the **Advanced Harmonic Balance** options dialog box before this measurement can be used. This checkbox is selected by default.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Dynamic Operating Point: OP_DYN

Summary

The OP_DYN measurement can be used to measure the dynamic operating point quantities. This measurement computes the operating point as a function of time. For small signal levels, this measurement generally give the same result as OP_DC. A typical use of this measurement might be to observe the change in gm over an operating cycle. The available operating point quantities can be selected using the **Measurement Component** drop-down list. Additional operating point quantities can be selected from the schematic browser by clicking the **...** button next to the **Measurement Component** drop-down.

Note: The **Time domain operating point** check box in the **Compute and save results from** area must be selected in the **Advanced Harmonic Balance** options dialog box before this measurement can be used. This checkbox is not selected by default.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. This measurement is invalid if the schematic contains more than one tone. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Amplitude Noise Special Density: AM_NOISE

Summary

AM_NOISE measures the spectral density of phase fluctuations of a noisy oscillator.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Offset Freq Index	Integer	0 to (NF-1)[b]
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in OSCNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

See the *AM_NOISE_F* documentation for details on this measurement. AM_NOISE_F displays results as a function of noise frequency, while AM_NOISE displays them as a function of arbitrary variable sweeps. Otherwise, the two are computationally the same.

Amplitude Noise Spectral Density (Vs. Offset Freq): AM_NOISE_F

Summary

AM_NOISE_F measures the spectral density of phase fluctuations of a noisy oscillator.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics[1]
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The x-axis for this measurement is the offset frequency range as defined by the OFstart and OFend parameters of the OSCNOISE element.

Computational Details

By conventional spot noise analysis a noisy oscillator waveform may be viewed as an amplitude and phase-modulated signal of the

form: $$x(t) = Re\left\{\left[A + Re\left\{a(\omega)e^{j\omega t}\right\}\right]\exp\left[j\left(\omega_c t + Re\left\{\Phi(\omega)e^{j\omega t}\right\}\right)\right]\right\}$$

where $\Phi(\omega)$ represents the random phasor of phase fluctuations at offset frequency ω; and $a(\omega)$ represents the random phasor of amplitude fluctuations at offset frequency ω.

AM_NOISE measures the amplitude noise to carrier ratio at offset frequency ω, given by the mean-square value of the random phasor $\frac{a(\omega)}{|A|}$,

$$S_a(\omega) = \frac{\langle a(\omega)a^*(\omega)\rangle}{|A|^2}$$

The above representation of a noisy oscillator signal focused on the fundamental component. The Harmonic Index parameter determines the output component of interest. In standard free-running oscillators Harmonic Index=1. In a multiplying-type oscillator, you should set the Harmonic Index to the appropriate multiple of the fundamental where amplitude noise is to be computed.

Phase noise L(fm) (vs. offset freq): LFM

Summary

LFM calculates the spectral density of phase fluctuations of the output voltage of a noisy oscillator. The expression used for this calculation is simplified compared to that used in PH_NOISE measurement but it may yield better agreement with the results calculated using other software packages.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

$$L(f_m) = \frac{\langle V_{-k} V^*_{-k} \rangle + \langle V_k V^*_k \rangle}{2 \langle V_{0,k} V_{0,k} \rangle}$$

where V_{-k} and V_k are the lower and upper noise sidebands, respectively, and $V_{0,k}$ is the harmonic component of the steady state solution.

SSB Noise-to-Carrier Ratio (Lower Sideband, Offset From Carrier): L_LSB

Summary

L_LSB measures the single-sideband noise-to-carrier ratio of a noisy oscillator, referenced to the lower sideband.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Offset Freq Index	Integer	0 to (NF-1)[b]
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in OSCNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

Please see PH_NOISE_F documentation for details on the difference between the USB and LSB phase noise measurements versus the PH_NOISE measurement.

SSB Noise-to-Carrier Ratio (Lower Sideband, Vs. Offset Freq): L_LSB_F

Summary

L_LSB_F measures the single-sideband noise-to-carrier ratio of a noisy oscillator, referenced to the upper sideband.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The x-axis for this measurement is the offset frequency range as defined by the OFstart and OFend parameters of the OSCNOISE element.

Computational Details

Please see PH_NOISE_F documentation for details on the difference between the USB and LSB phase noise measurements versus the PH_NOISE measurement.

SSB Noise-to-Carrier Ratio (Upper Sideband, Offset From Carrier): L_USB

Summary

L_USB measures the single-sideband noise-to-carrier ratio of a noisy oscillator, referenced to the upper sideband.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Offset Freq Index	Integer	0 to (NF-1)[b]
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in OSCNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

Please see PH_NOISE_F documentation for details on the difference between the USB and LSB phase noise measurements versus the PH_NOISE measurement.

SSB Noise-to-Carrier Ratio (Upper Sideband, Vs. Offset Freq): L_USB_F

Summary

L_USB_F measures the single-sideband noise-to-carrier ratio of a noisy oscillator, referenced to the upper sideband.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

Result

This measurement returns a real value. The x-axis for this measurement is the offset frequency range as defined by the OFstart and OFend parameters of the OSC-NOISE element.

Computational Details

Please see PH_NOISE_F documentation for details on the difference between the USB and LSB phase noise measurements versus the PH_NOISE measurement.

Oscillation Frequency: OSC_FREQ

Summary

OSC_FREQ displays an oscillator's frequency of oscillation determined by the OSCAPROBE element.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

.

Tuning Parameter: OSC_PARAM

Summary

This measurement is Obsolete with the swept variable capability introduced in 6.0.

OSC_PARAM displays the value of the tuning parameter in the tuning mode of oscillator analysis.

Parameters

Name	Type	Range
Data Source Name	Data source	Oscillators only

Result

This measurement returns a real value. The x-axis for this measurement is the value of the tuning parameter in MKS units

Graph Type

This measurement can be displayed on a Rectangular graph or Tabular grid.

Phase Noise Spectral Density (Offset from Carrier): PH_NOISE

Summary

PH_NOISE measures the spectral density of phase fluctuations of a noisy oscillator.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Offset Freq Index	Integer	0 to (NF-1)[b]

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

b NF is the number of noise frequency steps in OSCNOISE control.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Computational Details

Please see PH_NOISE_F documentation for details on this measurement. PH_NOISE_F displays results as a function of noise frequency, while PH_NOISE displays them as a function of arbitrary variable sweeps. Otherwise, the two are computationally the same.

Phase Noise Spectral Density: PH_NOISE_F

Summary

PH_NOISE_F measures the spectral density of phase fluctuations of a noisy oscillator.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Meter for noise meas	String	N/A
Harmonic Index[a]	Integer	-Max Harmonics to +Max Harmonics
Noise Computation Method	List of options	Default/Conversion Only

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value. The x-axis for this measurement is the offset frequency range as defined by the OFstart and OFend parameters of the OSCNOISE element..

Computational Details

By conventional spot noise analysis a noisy oscillator waveform may be viewed as an amplitude and phase-modulated signal of the form:

$$x(t) = Re\left\{\left[A + Re\left\{a(\omega)e^{j\omega t}\right\}\right]\exp\left[j\left(\omega_c t + Re\left\{\Phi(\omega)e^{j\omega t}\right\}\right)\right]\right\}$$

where $\Phi(\omega)$ represents the random phasor of phase fluctuations at offset frequency ω; and $a(\omega)$ represents the random phasor of amplitude fluctuations at offset frequency ω.

PH_NOISE measures the spectral density of phase fluctuations (in rad^2/Hz) at offset frequency ω, given by the mean-square value of the random phasor $\Phi(\omega)$,

$$S_{\phi}(\omega) = \langle \Phi(\omega)\Phi^{*}(\omega) \rangle$$

This representation of a noisy oscillator signal focused on the fundamental component. The **Harmonic Index** parameter determines the output component of interest. In standard free-running oscillators **Harmonic Index**=1. In a multiplying-type oscillator, you should set **Harmonic Index** to the appropriate multiple of the fundamental where phase noise is to be computed.

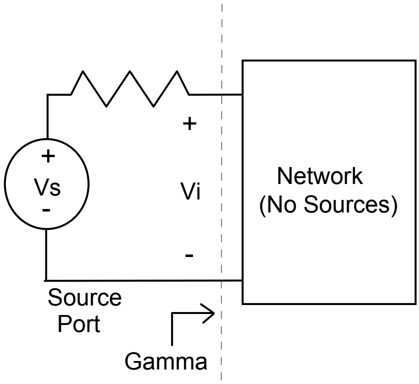
NOTE.

Many engineers loosely refer to the conventional noise-to-carrier ratio measurement as “phase noise”. PH_NOISE_F is a spectral density of phase noise and is not equivalent, though it is closely related, to the noise-to-carrier ratio. In particular, PH_NOISE_F is approximately 3 dB above noise-to-carrier ratio at near-carrier offsets. Use L_LSB_F and L_USB_F for more precise noise-to-carrier ratio measurements.

Large Signal Gamma: Gcomp

Summary

Gcomp can be used to measure a reflection coefficient under large signal excitation conditions. The reflection coefficient can be determined at any of the harmonic frequencies, although most commonly at the fundamental frequency. The measurement assumes that there are no sources looking in the direction in which the gamma is to be measured, as shown in the following figure.



Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

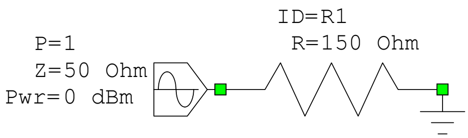
NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

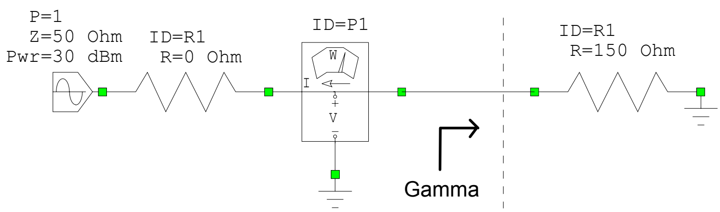
This measurement returns a complex value. The complex measurement can be displayed as a real value by specifying the magnitude, angle, real or imaginary component in the Add/Modify Measurement dialog box. The real value can also be displayed in dB by selecting the **dB** check box.

Computational Details

As an example, the large signal gamma for the following circuit would give gamma=$(Z_L - Z_0)/(Z_L + Z_0)$=(150-50)/(150+50)=0.5 for gamma measured at Port 1 and the fundamental frequency.



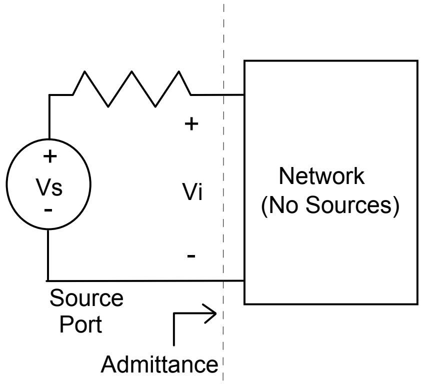
The power meter can also be used to measure the reflection coefficient. The large signal gamma measurement is normally used with the excitation port, and since the positive current flows into the port, the measurement automatically reverses the current direction for the computation (which requires the current to be defined as flowing out of the port). As a result, when using a power meter to measure the large signal gamma, the meter should be placed so the current path is the opposite of the actual current path flowing into the circuit. An example follows, where the large signal gamma measured at the meter P1 would be 0.5.



Large Signal Admittance: Ycomp

Summary

Ycomp can be used to measure admittance under large signal excitation conditions. The admittance can be determined at any of the harmonic frequencies, although most commonly at the fundamental frequency. The measurement assumes that there are no sources looking in the direction in which the admittance is to be measured, as shown in the following figure.



Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

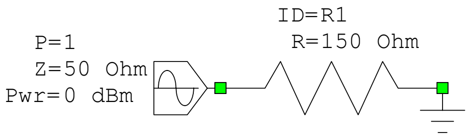
NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters

Result

This measurement returns a complex value in Conductance units. 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 real value can also be displayed in dB by selecting the **dB** check box.

Computational Details

As an example, the large signal admittance for the following circuit would give Y=1/150 for admittance measured at Port 1 and the fundamental frequency.

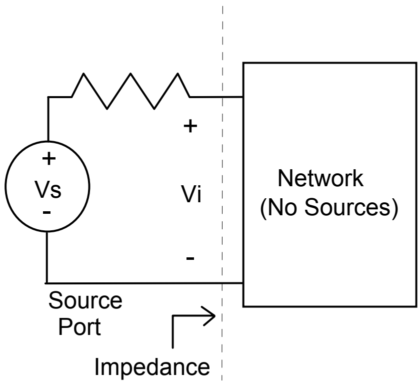


The power meter can also be used to measure the admittance. For more information, see the *Gcomp* measurement documentation.

Large Signal Impedance: Zcomp

Summary

Zcomp can be used to measure impedance under large signal excitation conditions. The impedance can be determined at any of the harmonic frequencies, although most commonly at the fundamental frequency. The measurement assumes that there are no sources looking in the direction in which the impedance is to be measured, as shown in the following figure.



Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

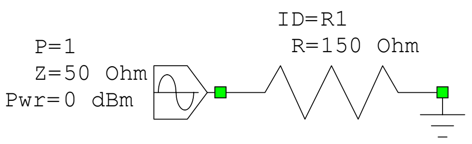
NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in Resistance units. 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 real value can also be displayed in dB by selecting the **dB** check box.

Computational Details

As an example, the large signal impedance for the following circuit would give Z=150 for impedance measured at Port 1 and the fundamental frequency.



The power meter can also be used to measure the impedance. For more information, see the *Gcomp* measurement documentation.

AM to AM of Fundamental: AMtoAM

Summary

AMtoAM is used to measure the AM to AM conversion of a circuit. The measurement will compute the magnitude of the power of the fundamental at the **Power Output Component** location. This measurement is identical to Pcomp with the **Harmonic Index** set to 1 and the complex modifier set to **Real** and **dB**. This measurement is not defined for multi-tone analysis. If doing multi-tone analysis, use the Pcomp measurement with the proper **Harmonic Index** set.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power Output Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in power in dBm.

AM to PM of Fundamental: AMtoPM

Summary

AMtoPM is used to measure the AM to PM conversion of a circuit. The measurement will compute the angle of the voltage of the fundamental at the **Voltage Output Component** location. This measurement is identical to Vcomp with the **Harmonic Index** set to 1 and the complex modifier set to **Angle**. This measurement is not defined for multi-tone analysis. If doing multi-tone analysis, use the Vcomp measurement with the proper **Harmonic Index** set.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Voltage Output Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in angle units.

Note

Voltage is used for this measurement because power at a purely real impedance (like 50 ohms) will have no imaginary part.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

DC to RF Efficiency: DCRF

Summary

DCRF computes the DC to RF conversion efficiency of a circuit. The DC power is computed from the total DC power of all sources in the circuit. The RF power is computed using the specified measurement element.

The DC to RF efficiency is defined as

$$DCRF = \left(\frac{P_{out}}{P_{dc}}\right) \cdot 100\%$$

where P_{out} is the power measured at the **Power Out Component** (usually the output port) and P_{dc} is the total DC power delivered by all the sources in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	NA
Power Out Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Input Mismatching Gain: INMG

Summary

INMG computes the nonlinear input mismatching gain of RF circuit blocks. The general equation for this implementation is:

$$INMG = \frac{\left|1 - \Gamma_{Source}\Gamma_{Zin}\right|^2}{(1 - \left|\Gamma_{Zin}\right|^2)(1 - \left|\Gamma_{Source}\right|^2)}$$

Where Γ_{Zin} represents the nonlinear input reflection coefficient of the RF circuit blocks and Γ_{Source} is the port impedance. This nonlinear behavior is generated due to the shift in operating conditions which the power sweep causes. The **Power in Component** must be measured at a port.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power In Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a scalar value of nonlinear input mismatching gain as a function of input power at a specified frequency.

Large Signal S-Parameter at Harmonic: LSSnm

Summary

LSSnm is used to compute the equivalent of an S-parameter under large signal excitation conditions. The computation requires that the "From" port be an excitation port. For example, the measurement of the complete set of S-parameters for a two port would require two separate simulations where one simulation places an excitation on port one (for measuring S_{21} and S_{11}) and the second simulation places and excitation on port two (for measuring S_{12} and S_{22}). This measurement also allows the selection of the harmonic frequency used for both the "From" port and the "To" port, enabling the measurement of quantities like conversion loss or gain in mixers and multipliers. This measurement can also be used to measure a large signal output return loss using two sources. The input port would be the large signal excitation of the circuit. The output port would be a tone 2 port with low power and a small offset from the fundamental. By picking the proper **Harmonic Index** for the “From” and “To” port, this will measure a large signal return loss at a port other than the excitation port.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	NA
Port (To)	String	N/A
Port (From)	String	N/A
Harmonic Index (To)[a]	Integer value	-Max Harmonics to +Max Harmonics
Harmonic Index (From)[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

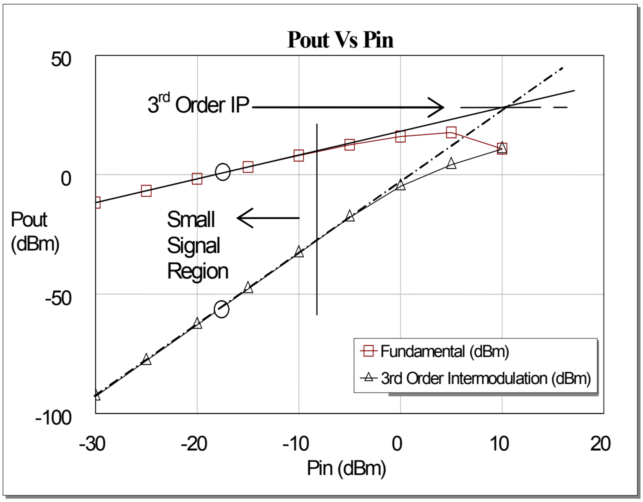
Result

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 real value can also be displayed in dB by selecting the **dB** check box.

Nth Order Intercept Point: OIPN

Summary

OIPN computes the nth-order output intercept point of a circuit, possibly having multi-tone excitation. 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 intercept point is given by

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

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. 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.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	NA
Output Power Meas. Component	String	N/A
Index of fund. comp.[a]	Integer	-Max Harmonics to +Max Harmonics
Index of IM comp.[a]	Integer	-Max Harmonics to +Max Harmonics
IP order	Integer	2 to 10

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in power in dBm.

Power Added Efficiency: PAE

Summary

PAE computes the power-added efficiency of a circuit. The DC power is computed from the total DC power of all sources in the circuit. The RF power is computed using the specified measurement element.

The power-added efficiency is defined as

$$PAE = \left(\frac{|P_{out}| - |P_{in}|}{P_{dc}}\right) \cdot 100\%$$

where P_{out} is the output power measured by the output measurement element (usually the output port), P_{in} is the input power delivered to the network, and P_{dc} is the total DC power delivered by all the sources in the circuit.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power In Component	String	N/A
Power Out Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Power Harmonic Component: Pcomp

Summary

Pcomp is used to measure a harmonic component of the power measured at a specified point in the circuit. The power value is returned as the complex magnitude of the RMS power component at the harmonic frequency.

To obtain the DC power, use a harmonic index of zero. To obtain the power at the fundamental frequency use a harmonic index of one.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in power units. 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 real value can also be displayed in dBm by selecting the **dBm** check box.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

FFT of Power for Specified Period: Pfft

Summary

Pfft calculates the power spectrum at the specified port or circuit component using FFT. It is intended to be used with transient simulators, such as HSPICE transient or Spectre transient.

You need to specify the Start and End Time, thus the fundamental period T is determined as T=End Time-Start Time, and the fundamental frequency f_0=1/T. The Start and End Time options allow skipping of the transient processes, so that FFT is applied to the portion of the transient waveform that corresponds to the steady state. The Start and End Time are specified in time units explicitly. Seconds (s), milliseconds (ms), microseconds (us), nanoseconds (ns), or picoseconds (ps) can be specified as units for Start and End Time.

The number of harmonics N specifies the highest frequency (Nf_0) in the calculated spectrum.

Since the FFT requires evenly spaced time samples while transient waveforms generated by HSPICE or Spectre have variable time step, interpolation of transient waveforms is performed. Two interpolation methods (Spline or Linear) can be selected. The default interpolation method (Spline) is recommended in most cases while Linear interpolation is used if the number of available time points in the transient waveform is extremely small. The latter condition occurs infrequently.

The oversampling factor improves accuracy of the calculated spectrum and avoids aliasing, thus the minimum allowed oversampling factor is 2 to avoid aliasing. The default value (4) is adequate in most cases.

Classic time domain windows can be applied to the waveform prior to calculating the FFT. The default is no windowing (this is optimal if the fundamental frequency is known).

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 Ports
Measurement Component	Port, Voltage source or a circuit element	N/A

Name	Type	Range
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, mn, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Number of Harmonics	Integer number	N>1. The highest frequency in the spectrum is N/(End Time-Start Time)
Interpolation method	String	Spline (recommended in most cases), or Linear
Oversampling factor	Integer number	2 to 16
Time domain window	String	Multiple classic time domain windows are available.

NOTE. All measurements have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns N+1 complex numbers (approximate Fourier components of the specified power). To perform this calculation, Approximate Fourier spectrum of current and voltage are calculated as described in Vfft and Ifft. The power spectrum is then calculated as:

$$P_k = \begin{cases} V_k I_k & \text{if} \quad k = 0(DC) \\ \frac{1}{2} V_k I^*{}_k & \text{if} \quad k \neq 0 \end{cases}$$

where N is the number of harmonics, and P_k (k=0, …, N) are the complex Fourier components of power. If dB display is selected, $10\log_{10}|P_k|$ is displayed.

Graph Type

This measurement can be displayed on a rectangular graph or table. Document frequency is one of the possible sweep variables.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Power Gain at Fundamental: PGain

Summary

PGain computes the transducer power gain and is the large-signal equivalent to the GT measurement. The transducer power gain is the ratio of the power delivered to the load to the power available from the source. The transducer power gain is given by

$$G_T = \frac{P_{\text{Power delivered to the load}}}{P_{\text{Power available from the source}}}$$

Both the input and output power are measured at the fundamental frequency.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power In Component	String	N/A
Power Out Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value.

Frequency Domain Power: Pharm

Summary

Pharm is used to measure the power spectrum. The power value is returned as the complex magnitudes of the RMS power components at each harmonic.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in power units. 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 real value can also be displayed in dB by selecting the **DB** check box. The x-axis for this measurement is always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Total Power: PT

Summary

PT is used to measure the total power delivered to the **Power Measurement Component**, including the power at DC and all harmonics. The total power is given by:

$$PT = \sum_h P(f_h)$$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in power units. The measurement can be displayed in dBm by selecting **dBm** as the **Result Type** in the Add/Modify Measurement dialog box.

Total Power in Band: PTB

Summary

PTB returns the total power in a frequency band from **Frequency Lower** to **Frequency Upper** entered in Hz.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Power Measurement Component	String	N/A
Frequency Lower (Hz)	Real value	0 to 1e+20
Frequency Upper (Hz)	Real value	0 to 1e+20

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in power units. The measurement can be displayed in dBm by selecting **dBm** as the **Result Type** in the Add/Modify Measurement dialog box.

Instantaneous Power: Ptime

Summary

Ptime is used to measure the instantaneous time domain power. The power value is returned as a real valued time waveform and is calculated by multiplying the instantaneous current and voltage values. For one-tone analysis, two periods of the waveform are displayed.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in power units. The measurement can be displayed in dBm by selecting **DBm** as the **Result Type** in the Add/Modify Measurement dialog box. The x-axis for this measurement is always in time units.

Voltage Harmonic Component: Vcomp

Summary

Vcomp is used to measure a harmonic component of the voltage measured at a point in the circuit as specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated. The voltage value is returned as the complex magnitude of the voltage component at the harmonic frequency (to obtain the r.m.s. value, you must divide by the square root of 2).

To obtain the DC voltage, use a harmonic index of zero (this will be the DC value with AC sources present, so it will capture self biasing). To obtain the voltage at the fundamental frequency use a harmonic index of one.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 real value can also be displayed in dB by selecting the **dB** check box which then displays 20*log10(|Val|).

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Differential Voltage Harmonic Component: VcompD

Summary

VcompD is used to measure a harmonic component of the voltage measured between two points in the circuit as specified by the **+Measurement Component** parameter and the **-Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage used will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage used will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated. The voltage value is returned as the complex magnitude of the voltage component at the harmonic frequency (to obtain the RMS. value, you must divide by the square root of 2).

To obtain the DC voltage, use a harmonic index of zero (this will be the DC value with AC sources present, so it will capture self biasing). To obtain the voltage at the fundamental frequency use a harmonic index of one.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A
Harmonic Index[a]	Integer value	-Max Harmonics to +Max Harmonics

a Multiple harmonic indices may appear in the Add/Modify Measurement dialog box depending on the number of tones associated with the simulation. See the *Microwave Office/Analog Office User Guide* for details.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 real value can also be displayed in dB by selecting the **dB** check box which then displays 20*log10(|Val|).

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

DC Voltage: VDC

Summary

VDC measures the DC voltage at a point in the circuit as specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the element does not have exactly two nodes, an error will be generated.

This DC value is a result of DC simulation only without the presence of any AC signals. Therefore, it will not show any self biasing effects. If you want to see self biasing, please use the Vcomp measurement with a harmonic index of 0.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units.

Differential DC Voltage: VDC_D

Summary

VDC_D measures the DC voltage between two points in the circuit as specified by the +Measurement Component parameter and the -Measurement Component parameter. If the measurement component parameter specifies a node, then the voltage used will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage used will be the voltage across this element (Vnode1-Vnode2). If the element does not have exactly two nodes, an error will be generated.

This DC value is a result of DC simulation only without the presence of any AC signals. Therefore, it will not show any self biasing effects. If you want to see self biasing, please use the VcompD measurement with a harmonic index of 0.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units.

Voltage Envelope: Venv

Summary

Venv displays the complex envelope of a voltage waveform in the circuit as specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the element does not have exactly two nodes, an error will be generated.

It is often convenient to think of signals in nonlinear circuits as

$$x(t) = \sum_{n=0}^{N} Re\left\{\tilde{x}_n(t)e^{jn\omega_c t}\right\}$$

Venv displays the component of most interest, $\tilde{x}_1(t)$

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
No. Samples	Integer value	N/A
No. Periods	Integer value	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 x-axis for this measurement is always in time units.

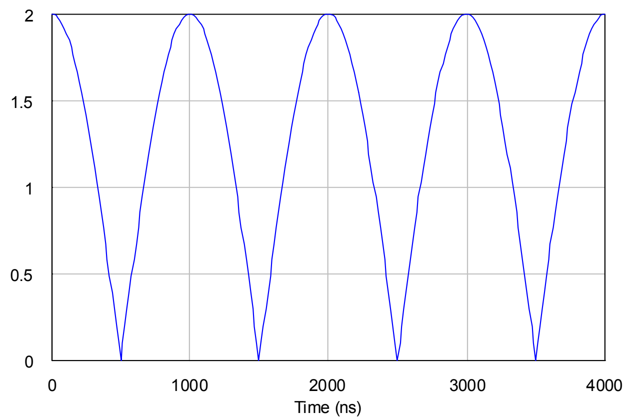
Parameters

No. Samples may be used to refine display resolution. Default is 128 which should be sufficient for the majority of applications. If the specified number of samples is smaller than the number of frequency components comprising the envelope in the HB simulation, the **No. Samples** specification is ignored.

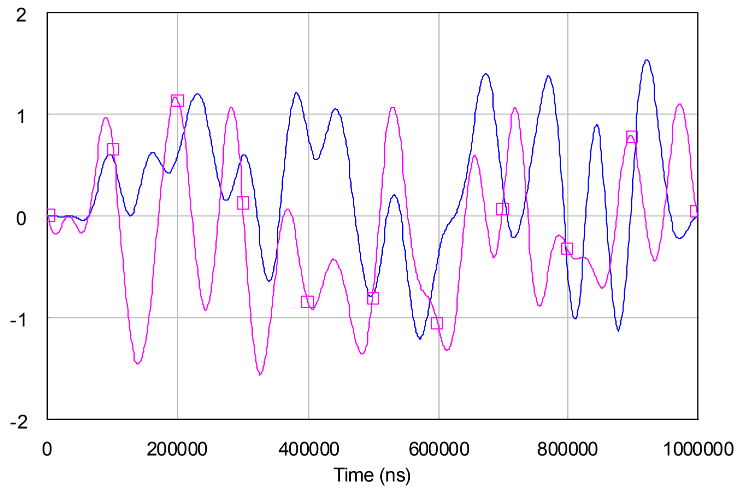
The envelope waveform is repeated **No. Periods** times. This may be useful for two-tone signals where the modulation is sinusoidal.

Example output is shown below.

Example 1: Magnitude of the voltage envelope for a two-tone signal. The tones are spaced by 1 MHz and the available power is 10 dBm per tone. The voltage is measured across a matched 50 Ohm load. **No. Periods** is equal to 2, remaining parameters are at default values.



Example 2: The I/Q components of the signal file PI4-DQPSK.sig Available signal power is 10 dBm, the load is matched.



Voltage Eye Diagram: Veye

Summary

Veye displays the time domain voltage in the form of an eye diagram at a point in the circuit as specified by the Measurement Component parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the element does not have exactly two nodes, an error will be generated. This measurement is applicable only to circuits that are driven by PORT_ARBS, PORT_PRBS and PORT_SIG elements. Please refer to PORT_SIG documentation for signal file modifications needed to accommodate eye diagram displays.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

Differential Voltage Eye Diagram: VeyeD

Summary

VeyeD displays the time domain voltage in the form of an eye diagram at a point in the circuit as specified by the +Measurement Component and -Measurement Component parameters. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the element does not have exactly two nodes, an error will be generated. This measurement is applicable only to circuits that are driven by PORT_ARBS, PORT_PRBS and PORT_SIG elements. Please refer to PORT_SIG documentation for signal file modifications needed to accommodate eye diagram displays.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

FFT of Voltage for Specified Period: Vfft

Summary

Vfft calculates the spectrum of the specified periodic voltage waveform using FFT. This measurement is intended to be used with transient simulators, such as HSPICE transient or Spectre transient.

You need to specify the Start and End Time, thus the fundamental period T is determined as T=End Time-Start Time, and the fundamental frequency f_0=1/T. The Start and End Time options allow skipping of the transient processes, so that FFT is applied to the portion of the transient waveform that corresponds to the steady state. The Start and End Time are specified in time units explicitly. Seconds (s), milliseconds (ms), microseconds (us), nanoseconds (ns), or picoseconds (ps) can be specified as units for Start and End Time.

The number of harmonics N specifies the highest frequency (Nf_0) in the calculated spectrum.

Since the FFT requires evenly spaced time samples while transient waveforms generated by HSPICE or Spectre have variable time step, interpolation of transient waveforms is performed. Two interpolation methods (Spline or Linear) can be selected. The default interpolation method (Spline) is recommended in most cases while Linear interpolation is used if the number of available time points in the transient waveform is extremely small. The latter condition occurs infrequently.

The oversampling factor improves accuracy of the calculated spectrum and avoids aliasing. Thus, the minimum allowed oversampling factor is 2 to avoid aliasing. The default value (4) is adequate in most cases.

Classic time domain windows can be applied to the waveform prior to calculating the FFT. The default is no windowing (this is optimal if the fundamental frequency is known).

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port, voltage probe, or a terminal of a circuit element	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field.
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Number of harmonics	Integer number	N>1. The highest frequency in the spectrum is N/(End Time-Start Time)
Interpolation method	String	Spline (recommended in most cases), or Linear
Oversampling factor	Integer number	2 to 16
Time domain window	String	Multiple classic time domain windows are available.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns N+1 complex numbers (approximate Fourier components of the specified voltage). The notation is

$$v(t) = V_0 + Re\left[\sum_{k=1}^{N} V_k \exp[jk\omega_0 t]\right]$$

where N is the number of harmonics, and V_k (k=0, …, N) are the complex Fourier components. If dB display is selected, $20\log_{10}|V_k|$ is displayed.

Graph Type

This measurement can be displayed on a rectangular graph or table. Document frequency is one of the possible sweep variables.

If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

FFT of Differential Voltage for Specified Period: VfftD

Summary

VfftD calculates the spectrum of a differential voltage. This measurement is intended to be used with transient simulators, such as HSPICE transient or Spectre transient.

You need to specify the Start and End Time, thus the fundamental period T is determined as T=End Time-Start Time, and the fundamental frequency f_0=1/T. The Start and End Time options allow skipping of the transient processes, so that FFT is applied to the portion of the transient waveform that corresponds to the steady state. The Start and End Time are specified in time units explicitly. Seconds (s), milliseconds (ms), microseconds (us), nanoseconds (ns), or picoseconds (ps) can be specified as units for Start and End Time.

The number of harmonics N specifies the highest frequency (Nf_0) in the calculated spectrum.

Since the FFT requires evenly spaced time samples while transient waveforms generated by HSPICE or Spectre have variable time step, interpolation of transient waveforms is performed. Two interpolation methods (Spline or Linear) can be selected. The default interpolation method (Spline) is recommended in most cases while Linear interpolation is used if the number of available time points in the transient waveform is extremely small. The latter condition occurs infrequently.

The oversampling factor improves accuracy of the calculated spectrum and avoids aliasing. Thus the minimum allowed oversampling factor is 2 to avoid aliasing. The default value (4) is adequate in most cases.

Classic time domain windows can be applied to the waveform prior to calculating the FFT. The default is no windowing (this is optimal if the fundamental frequency is known). Parameters

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port, voltage probe, or a terminal of a circuit element	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field.
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Number of harmonics	Integer number	N>1. The highest frequency in the spectrum is N/(End Time-Start Time)
Interpolation method	String	Spline (recommended in most cases), or Linear
Oversampling factor	Integer number	2 to 16
Time domain window	String	Multiple classic time domain windows are available.

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns N+1 complex numbers (approximate Fourier components of the specified voltage). The notation is:

$$v(t) = V_0 + Re\left[\sum_{k=1}^{N} V_k \exp[jk\omega_0 t]\right]$$

where N is the number of harmonics, and V_k (k=0, …, N) are the complex Fourier components. If dB display is selected, $20\log_{10}|V_k|$ is displayed.

Graph Type

This measurement can be displayed on a rectangular graph or table. Document frequency is one of the possible sweep variables.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Voltage Gain: Vgain

Summary

Vgain measures the voltage gain between two points in the circuit as specified in **Voltage In Component** and **Voltage Out Component**. **Voltage In Component** and **Voltage Out Component** can be any node, port, element terminal, source, or measurement device. The gain value is returned as a ratio of complex magnitude of output voltage to the input voltage at a given harmonic frequency.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Voltage in Component	String	N/A
Voltage in Harmonic Index (0 GHz)	Integer	-Max Harmonics to +Max Harmonics
Voltage Out Component	String	N/A
Voltage Out Harmonic Index (0 GHz)	Integer	-Max Harmonics to +Max Harmonics

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unit less complex value. The complex measurement can be displayed as a real value by specifying the magnitude (**Mag.**), **Angle**, **Real** or imaginary (**Imag.**) component in the Add/Modify Measurement dialog box. The real value can also be displayed in dB by selecting the **dB** checkbox which then displays 20*log10(|Val|). The x-axis for this measurement is always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Differential Voltage Gain: VgainD

Summary

VgainD measures the voltage gain between differential input voltage and differential output voltage. Two points in the circuit specified in **+Voltage In Component** and **-Voltage In Component** measure the input voltage where as points specified in **+Voltage Out Component** and **-Voltage Out Component** measure the output voltage. Voltage In and Voltage Out components can be any node, port, element terminal, source, or a measurement device. The gain value is returned as a ratio of complex magnitude of output voltage to the input voltage at a given harmonic frequency.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Voltage In Component	String	N/A
-Voltage In Component	String	N/A
Voltage in Harmonic Index (0 GHz)	Integer	-Max Harmonics to +Max Harmonics
+Voltage Out Component	String	N/A
-Voltage Out Component	String	N/A
Voltage Out Harmonic Index (0 GHz)	Integer	-Max Harmonics to +Max Harmonics

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unit less complex value. The complex measurement can be displayed as a real value by specifying the magnitude (**Mag.**), **Angle**, **Real** or imaginary (**Imag.**) component in the Add/Modify Measurement dialog box. The real value can also be displayed in dB by selecting the **dB** checkbox which then displays 20*log10(|Val|). The x-axis for this measurement is always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Frequency Domain Voltage: Vharm

Summary

Vharm is used to measure the voltage spectrum at a point in the circuit as specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated. The voltage value is returned as a spectrum of complex magnitudes of all the voltage components at each harmonic frequency (to obtain the RMS. value, you must divide by the square root of 2).

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 real value can also be displayed in dB by selecting the **DB** check box which then displays 20*log10(|Val|). The x-axis for this measurement is always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Differential Frequency Domain Voltage: VharmD

Summary

VharmD is used to measure the voltage spectrum between two points in the circuit as specified by the **+Measurement Component** parameter and the **-Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage used will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage used will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated. The voltage value is returned as a spectrum of complex magnitudes of all the voltage components at each harmonic frequency (to obtain the RMS value, you must divide by the square root of 2).

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units. 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 real value can also be displayed in dB by selecting the **DB** check box which then displays 20*log10(|Val|). The x-axis for this measurement is always in frequency units.

CAUTIONARY NOTE. If a sinusoidal nonlinear source (e.g. PORT1 or ACVS) has the parameter Ang=0 and is ideally terminated, then a nonlinear measurement like this one made at the fundamental output of that source will have an angle of -

90deg. This discrepancy is consistent with the definition of the sine wave sources (as in SPICE), and the Fourier-based harmonic component measurements. To avoid confusion, always measure or calculate the gain, and plot its angle.

Voltage Spectrum Calculated from a Transient Simulation of an Autonomous Circuit: Vspec

Summary

Vspec calculates voltage spectrum from transient simulations (HSPICE or Spectre) for an autonomous circuit such as an oscillator. The measurement is similar in purpose to Vharm with the main difference being that it determines the fundamental frequency of the waveform. The measurement is therefore intended to be used with autonomous circuits where the fundamental frequency is not known beforehand. Although it is possible to use this measurement for driven circuits, this is not recommended as Vharm will provide a more accurate result where the fundamental frequency is known.

The voltage value is returned as a spectrum of complex magnitudes of all the voltage components at each harmonic frequency (to obtain the r.m.s. value, divide by $\sqrt{2}$).

To obtain a meaningful result the transient simulation needs to be run until steady state is reached. Vspec first determines the approximate fundamental frequency by starting from the end of the waveform and finding the last two time values where the signal rises to cross the DC average. By default, VSpec then uses the last 10 periods of this approximate fundamental frequency to calculate the spectrum. The PERCENT OF DATA parameter sets how much of this data to use (starting from the end). For example, if it is set to 20%, only the last 20% of the transient waveform (two periods) will be examined, and the leading 80% will be discarded,

The spectrum can be calculated using FFT or least square approximation with harmonic functions (“Fit”). Use the SPECTRUM COMPUTATION METHOD parameter to choose. The FFT approach is faster while Fit can be more accurate, so long as the NUMBER OF HARMONICS is set large enough to include all significant harmonics in the signal. For example, assume the spectrum of the signal contains N significant harmonics (and harmonics beyond that can be neglected within specified error tolerance). Then for the FFT method you can specify the number of harmonics k<N and still get the right answer for those k harmonics. If the Fit approach is used, you need to specify at least N harmonics to get the right answer, otherwise the amplitude of these k harmonics will come out incorrect. Specifying more harmonics than are needed for the Fit approach will not affect the accuracy, but will increase the calculation time.

The user specifies the number of harmonics N to use. The voltage is approximated as

$$V(t) = V_0 + Re\left[\sum_{n=1}^{N} V_n \exp(j\omega_0 nt)\right] \quad (1)$$

If FFT is chosen, the OVERSAMPLING FACTOR (m) and the TIME DOMAIN WINDOW parameters are used. The number of time samples used for FFT is (N+1)m. Thus oversampling factor increases the accuracy of FFT. The minimum allowed value of the oversampling factor is 2, and the maximum is 32. There is usually no benefit in increasing m beyond m=8. The default is not to use any time domain window. The usage of classic time domain windows is beneficial only if at least 10 periods of the fundamental are available. No time domain window will be used if less then 4 periods of the fundamental frequency are available.

Parameters

Name	Type	Range
Data Source Name	Schematic	N/A
Measurement Component	Port or voltage probe	N/A
Number of Harmonics	Integer Value	2 to 512, default value of 10
Spectrum Computation Method	String	FFT or Fit
Oversampling factor for FFT	Integer Value	2 to 32, recommended value is 4
Time Domain Window	String	Classic time domain spectral windows such as Hamming, Hann, etc.
Percent of data to use	Real Value	1 to 100, default of 100

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a complex value in voltage units like the Vharm measurement. See the Vharm documentation for more info.

Notes

This measurement is intended and can only be used with transient simulations, either HSPICE transient of Spectre transient. It will not work with Harmonic Balance, and an error message displays to this effect. Use Vharm measurement with Harmonic Balance.

At present this measurement works with one tone only, as shown by Eqn. (1). For example, it will not yield correct results for AM signals as it will use the carrier frequency for fundamental frequency.

Time Domain Voltage: Vtime

Summary

Vtime is used to measure a time domain voltage at a point in the circuit as specified by the **Measurement Component** parameter. If the measurement component parameter specifies a node, then the voltage measured will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage measured will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated.

For one-tone analysis, two periods of the waveform will be displayed. For multi-tone analysis, many time samples may be generated in order to accurately render the waveform. The maximum number of time samples is configurable, refer to the Advanced HB options dialog.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
Measurement Component	String	N/A
Offset	List of options	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Computational Details

The **Offset** setting controls any desired shift in the waveform. "None" will introduce no shift. "First Point" will offset by the first waveform point value so the first point will always be 0. "Average" will offset by the average value of the waveform. "RMS" will offset by the Root Mean Square (RMS) value of the waveform.

Differential Time Domain Voltage: VtimeD

Summary

VtimeD is used to measure a time domain voltage between two points in the circuit as specified by the +Measurement Component parameter and the -Measurement Component parameter. If the measurement component parameter specifies a node, then the voltage used will be the voltage at this node referenced to ground. If the measurement component parameter specifies an element with two nodes, then the voltage used will be the voltage across this element (Vnode1-Vnode2). If the specified element does not have exactly two nodes, an error will be generated.

For one-tone analysis, two periods of the waveform will be displayed. For multi-tone analysis, many time samples may be generated in order to accurately render the waveform. The maximum number of time samples is configurable, refer to the Advanced HB options dialog.

Parameters

Name	Type	Range
Data Source Name	Subcircuit	N/A
+Measurement Component	String	N/A
- Measurement Component	String	N/A
Offset	List of options	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the Microwave Office/Analog Office User Guide for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Computational Details

The **Offset** setting controls any desired shift in the waveform. “None” will introduce no shift. “First Point” will offset by the first waveform point value so the first point will always be 0. “Average” will offset by the average value of the waveform. “RMS” will offset by the Root Mean Square (RMS) value of the waveform.

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Eye Width	Percent	5% to 40%, default 20%

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Differential Eye Amplitude: Eye_AmplitudeD

Summary

Eye_AmplitudeD computes the amplitude metric of an eye diagram of a differential voltage signal. 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 Differential Eye Level Info measurement Eye_LevelD.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
- Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Eye Width	Percent	5% to 40%, default 20%

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

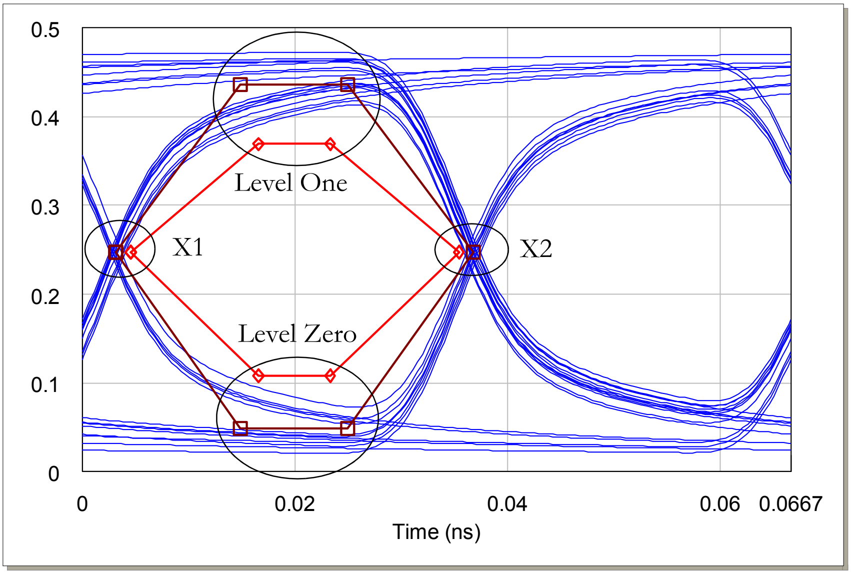
When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

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 Voltage eye diagram measurement ***Veye*** 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. **Crossing Display** 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. **Level Display** 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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of level crossing time on a sweep variable. Document frequency is one of the possible sweep variables.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

Differential Eye Corners: Eye_CornersD

Summary

Eye_CornersD plots the two eye crossings (X1 and X2), the Level One edges and the Level Zero edges of an eye diagram of a differential voltage signal.

Eye_CornersD is identical to the Eye_Corners measurement except it measures differential voltage rather than absolute voltage. See Eye_Corners for details on the measurement.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value in voltage units. The x-axis for this measurement is always in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of level crossing time on a sweep variable. Document frequency is one of the possible sweep variables.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

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 Voltage Eye Diagram measurement *Veye* can be used to visualize the eye crossings on an eye diagram.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

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

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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** value, 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**, 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** 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 **Trace Width** should normally be set so the eye diagram contains two to three crossing points.

Differential Eye Crossing Info: Eye_CrossingD

Summary

Eye_CrossingD locates the left (X1) or right (X2) crossing point of an eye diagram of a differential voltage signal 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 Differential Eye Corners measurement *Eye_CornersD* along with the Differential Voltage Eye Diagram measurement *VeyeD* can be used to visualize the eye crossings on an eye diagram.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

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

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

Eye_CrossingD is identical to the Eye Crossing Info measurement Eye_Crossing except it measures differential voltage rather than absolute voltage. See *Eye_Crossing* for details on the measurement.

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time

Name	Type	Range
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unitless real value.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

Differential Eye Extinction Ratio: Eye_ExtRatioD

Summary

Eye_ExtRatioD computes an extinction ratio or percentage for an eye diagram of a differential voltage signal. 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 Differential Eye Level Info measurement *Eye_LevelD*.

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

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value

Name	Type	Range
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unitless real value.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

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. You can change these defaults 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 *Eye_Transition*.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
*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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments do not line up properly.

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

Differential Eye Fall Time: Eye_FallTimeD

Summary

Eye_FallTimeD 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. You can change these defaults 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 *Eye_Transition*.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
*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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Eye Height: Eye_Height

Summary

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

$$\begin{aligned} Height = & (Level\_One\_mean - 3 \cdot Level\_One\_sigma) \\ & - (Level\_Zero\_mean + 3 \cdot Level\_Zero\_sigma) \end{aligned}$$

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are $%Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Eye Width	Percent	5% to 40%, default 20%

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Differential Eye Height: Eye_HeightD

Summary

Eye_HeightD computes the height metric of an eye diagram of a differential voltage signal. The height metric is defined as:

$$\begin{aligned} Height &= (Level\_One\_mean - 3 \cdot Level\_One\_sigma) \\ &\quad - (Level\_Zero\_mean + 3 \cdot Level\_Zero\_sigma) \end{aligned}$$

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 Differential Eye Level Info measurement Eye_LevelD.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are $%Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Eye Width	Percent	5% to 40%, default 20%

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Eye Inverse Extinction Ratio: Eye_InvExtRatio

Summary

Eye_InvExtRatio computes an inverse 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. *Max*(*Y*) is the maximum 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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time

Name	Type	Range
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unitless real value.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Differential Eye Inverse Extinction Ratio: Eye_InvExtRatioD

Summary

Eye_InvExtRatioD computes an inverse extinction ratio or percentage for an eye diagram of a differential voltage signal. 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. *Max*(*Y*) is the maximum Y value of the signal. The computation of these values is performed in a manner similar to the Differential Eye Level Info measurement *Eye_LevelD*.

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

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value

Name	Type	Range
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%
Output Type	List of Options	N/A

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a unitless real value.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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* measurement along with the Voltage Eye Diagram measurement *Veye* can be used to visualize the levels on an eye diagram.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%
Eye Window Center (%)	Percent	10.5% to 89.5%, default 50%

Name	Type	Range
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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value whose units depend on the **Output Type**. For the mean, sigma, peak, maximum Y and minimum Y options the units are voltage. For the count options the value is unitless.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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** 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.

Differential Eye Level Info: Eye_LevelD

Summary

Eye_LevelD displays information about the Level One or Level Zero points of an eye diagram of a differential voltage signal. 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**.

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 Differential Eye Corners measurement *Eye_CornersD* along with the Differential Voltage Eye Diagram measurement *VeyeD* can be used to visualize the eye crossings on an eye diagram.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value

Name	Type	Range
Max Traces	Integer value	0 to number of symbols in sequence
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
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value whose units depends upon the **Output Type**. For the mean, sigma, peak, maximum Y and minimum Y options the units are voltage. For the count options the value is unitless.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Eye_LevelD is identical to the Eye Level Info measurement except it measures differential voltage rather than absolute voltage. See *Eye_Level* for details.

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Differential Eye Overshoot: Eye_OvershootD

Summary

Eye_OvershootD computes the overshoot of Level One (the upper level) of an eye diagram of a differential voltage signal. 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 Differential Eye Level Info measurement *Eye_LevelD*.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Differential Eye Q Factor: Eye_QFactorD

Summary

Eye_QFactorD computes a Q factor for an eye diagram of a differential voltage signal. The Q factor is computed as:

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

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

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
*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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Differential Eye Rise Time: Eye_RiseTimeD

Summary

Eye_RiseTimeD computes the rise time for an eye diagram of a differential voltage signal. The rise time 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
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
*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. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they

change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

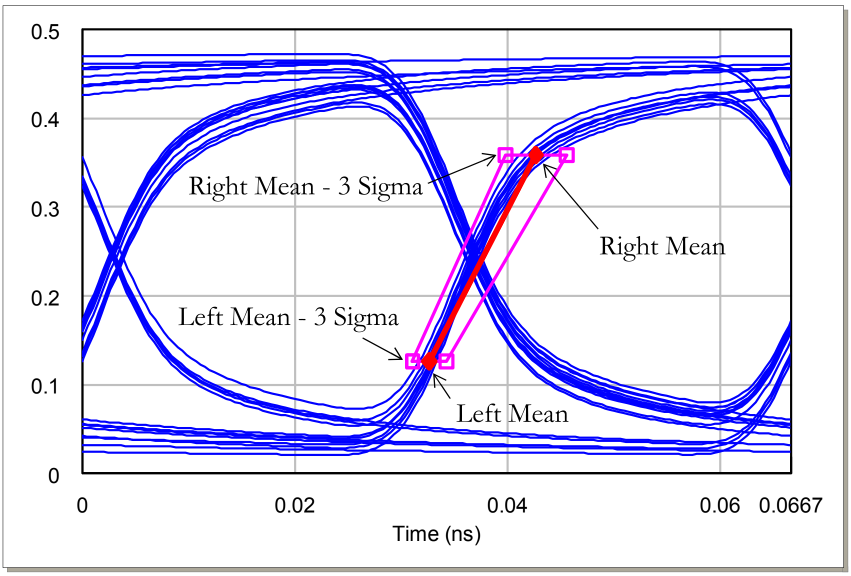
Eye Transition: Eye_Transition

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 the previous 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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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%
*Y Crossing Level	Voltage	Unlimited
*Peak Smoothing	Percent	0% to 50%

Name	Type	Range
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

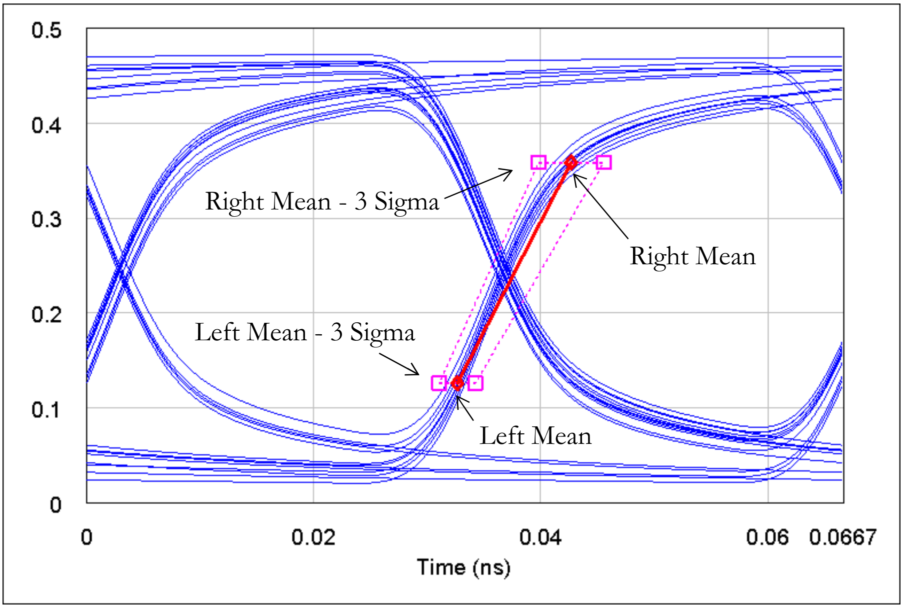
Differential Eye Transition: Eye_TransitionD

Summary

Eye_TransitionD displays information about the transitions between Level One and Level Zero of an eye diagram of a differential voltage signal. 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 the previous 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 displays.

Eye_TransitionD 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
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
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%
*Y Crossing Level	Voltage	Unlimited

Name	Type	Range
*Peak Smoothing	Percent	0% to 50%
*Peak Threshold	Percent	0% to 100%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

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 voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

Differential Eye Undershoot: Eye_UndershootD

Summary

Eye_UndershootD computes the undershoot of Level Zero (the lower level) of an eye diagram of a differential voltage signal. 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 Differential Eye Level Info measurement *Eye_LevelD*.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
-Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	0 to number of symbols in sequence
Eye Window Width (%)	Percent	1% to 80%, default 20%

** indicates a secondary parameter*

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. See the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of voltage.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width**. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, the individual segments do not line up properly.

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

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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Threshold	Not used	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Differential Eye Width: Eye_WidthD

Summary

Eye_WidthD computes the width metric of an eye diagram of a differential voltage signal. 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 Differential Eye Crossing Info measurement *Eye_CrossingD*.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or Voltage probe	N/A
- Measurement Component	Port or Voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Threshold	Not used	0.05 to 0.4, default 0.2

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Jitter Measurement: 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
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence
Threshold	Not used	N/A

Name	Type	Range
Method	String	Peak-Peak or RMS

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Differential Jitter Measurement: JitterD

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 Differential Eye Crossing Info measurement Eye_CrossingD.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Trace Width	Integer value	1 to number of symbols in sequence if units are Symbols or 0 to max time if units are time
Trace Units	List of Options	Symbols or discrete time value
Delay	Number	Unlimited if units are %Symbols or max time if units are time
Delay Units	List of Options	%Symbols or discrete time value
Max Traces	Integer value	1 to number of symbols in sequence

Name	Type	Range
Threshold	Not used	N/A
Method	String	Peak-Peak or RMS

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the "Swept Parameter Analysis" chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters.

Result

This measurement returns a real value with units of time.

Graph Type

This measurement is normally displayed in a rectangular graph, in a table, or used in Output Equations.

Notes

When working with eye diagrams, the simulator calculates the complete time waveform and then cuts it into segments based on the **Trace Width** setting. To obtain a proper eye diagram, this parameter should be an integer if the unit type is **Symbols**, or set to multiples of the symbol period if the unit type is set to a time value. If not, then the individual segments will not line up properly.

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

Overshoot Voltage: Overshoot

Summary

The Overshoot measurement calculates the overshoot voltage defined as the difference between the maximum voltage and the high voltage reference (logical "1" level) HighRef.

This measurement reports the first overshoot value found within the user-specified time interval, and zero if there is no overshoot.

The HighRef and LowRef voltages can be either determined automatically (default behavior), or user-specified in Volts. In the latter case, you need to select the **Specify** checkbox.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage in volts	Can be user-specified when Specify checkbox is selected, or ignored if the checkbox is cleared.
Specify	Boolean value	Selected or cleared

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Overshoot Voltage: OvershootD

Summary

The OvershootD measurement calculates the overshoot voltage defined as the difference between the maximum voltage and the high voltage reference (logical "1" level) HighRef.

This measurement reports the first overshoot value found within the user-specified time interval, and zero if there is no overshoot.

The HighRef and LowRef voltages can be either determined automatically (default behavior), or user-specified in Volts. In the latter case, you need to select the **Specify** checkbox.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage in volts	Can be user-specified when Specify checkbox is selected, or ignored if the checkbox is cleared.
Specify	Boolean value	Selected or cleared

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Time of the Level Crossing: Tcross

Summary

Tcross calculates the time of level crossing at the user-specified voltage level, either on the Rising or Falling edge. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators.

The Start and End Time are specified in time units explicitly. The Level needs to be explicitly specified in voltage units. You can also specify the crossing number to get the time of nth crossing at the specified edge in the specified time interval. If the actual number of crossings is less than n, the measurement issues a warning message and returns DBL_MAX=1.7976931348623158e+308.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage	Between low and high voltage level to get an answer
Edge	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of level crossing time on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Time of the Level Crossing: TcrossD

Summary

TcrossD calculates the time of level crossing at the user-specified voltage level, either on the Rising or Falling edge. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators.

The Start and End Time are specified in time units explicitly. The Level needs to be explicitly specified in voltage units. You can also specify the crossing number to get the time of nth crossing at the specified edge in the specified time interval. If the actual number of crossings is less than n, the measurement issues a warning message and returns DBL_MAX=1.7976931348623158e+308.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage	Between low and high voltage level to get an answer
Edge	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of level crossing time on a sweep variable. Document frequency is one of the possible sweep variables.

Transition Time: Ttime

Summary

Ttime calculates the transition (Rise or Fall) time for a time domain voltage waveform. This measurement can be used with any simulator that generates time domain waveform, i.e. harmonic balance, HSPICE transient, or Spectre transient. The intended application is for waveforms that represent digital signals (i.e. pulses). You can specify the transition type (rise or fall), the time interval in which to analyze (in the units of time), and the reference levels LowRef and HighRef. If there are several transitions in the specified time interval the measurement reports the average, minimum, or maximum transition time based on the setting for the Averaging Method.

The rise time is defined as the time it takes the signal to transition from LowRef (vL) to HighRef (vH) on the rising edge. The fall time is defined as the time it takes the signal to transition from HighRef to LowRef on the falling edge. LowRef and HighRef can be specified explicitly in voltage units, or in percents of the voltage span between Low and High voltage.

The Low voltage (bottom reference level, or baseline) is defined as the level at which the signal settles for the logical "0" level. Note that it is not necessarily the absolute minimum voltage, as there can be undershoot. The High voltage (top reference level, or topline) is defined as the level at which the signal settles for the logical "1" level. Similarly, it is not necessarily the absolute maximum voltage, as there can be overshoot.

If the LowRef and Highref are specified as percentages, the code automatically determines the Low and High reference levels based on the histogram method. The entire available waveform is used to determine Low and High reference levels.

If this measurement is used for waveforms other than pulses, you should specify the low and high voltage level explicitly, not as a percentage of the span between LowRef and HighRef, as these voltages will not be well defined.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling

Name	Type	Range
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Low Level	Real value	LowRef, specified as percentage, or in voltage units (volts, millivolts, or microvolts) depending on the corresponding Units field.
High Level	Real value	HighRef, specified as percentage, or in voltage units (volts, millivolts, or microvolts) depending on the corresponding Units field.
Averaging method	String	Average, Minimum, or Maximum

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Transition Time: TtimeD

Summary

TtimeD calculates the transition (Rise or Fall) time for a time domain voltage waveform. This measurement can be used with any simulator that generates time domain waveform, i.e. harmonic balance, HSPICE transient, or Spectre transient. The intended application is for waveforms that represent digital signals (i.e. pulses). You can specify the transition type (rise or fall), the time interval in which to analyze (in the units of time), and the reference levels LowRef and HighRef. If there are several transitions in the specified time interval the measurement reports the average, minimum, or maximum transition time based on the setting for the Averaging Method.

The rise time is defined as the time it takes the signal to transition from LowRef (vL) to HighRef (vH) on the rising edge. The fall time is defined as the time it takes the signal to transition from HighRef to LowRef on the falling edge. LowRef and HighRef can be specified explicitly in voltage units, or in percents of the voltage span between Low and High voltage.

The Low voltage (bottom reference level, or baseline) is defined as the level at which the signal settles for the logical "0" level. Note that it is not necessarily the absolute minimum voltage, as there can be undershoot. The High voltage (top reference level, or topline) is defined as the level at which the signal settles for the logical "1" level. Similarly, it is not necessarily the absolute maximum voltage, as there can be overshoot.

If the LowRef and Highref are specified as percentages, the code automatically determines the Low and High reference levels based on the histogram method. The entire available waveform is used to determine Low and High reference levels.

If this measurement is used for waveforms other than pulses, you should specify the low and high voltage level explicitly, not as a percentage of the span between LowRef and HighRef, as these voltages will not be well defined.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A

Name	Type	Range
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Low Level	Real value	LowRef, specified as percentage, or in voltage units (volts, millivolts, or microvolts) depending on the corresponding Units field.
High Level	Real value	HighRef, specified as percentage, or in voltage units (volts, millivolts, or microvolts) depending on the corresponding Units field.
Averaging method	String	Average, Minimum, or Maximum

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Undershoot Voltage: Undershoot

Summary

The Undershoot measurement calculates the undershoot voltage defined as the difference between the minimum voltage and the low voltage reference (the logical "0" level) LowRef. See the *Ttime* measurement for the definition of HighRef and LowRef.

This measurement reports the first undershoot value found within the user-specified time interval, and zero if there is no undershoot.

The HighRef and LowRef voltages can be either determined automatically (default behavior), or user-specified in Volts. In the latter case, you need to select the **Specify** checkbox.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage in volts	Can be user-specified when Specify checkbox is selected, or ignored if the checkbox is cleared.
Specify	Boolean value	Selected or cleared

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Undershoot Voltage: UndershootD

Summary

The UndershootD measurement calculates the undershoot voltage defined as the difference between the minimum voltage and the low voltage reference (the logical "0" level) LowRef. See the *Ttime* measurement for the definition of HighRef and LowRef.

This measurement reports the first undershoot value found within the user-specified time interval, and zero if there is no undershoot.

The HighRef and LowRef voltages can be either determined automatically (default behavior), or user-specified in Volts. In the latter case, you need to select the **Specify** checkbox.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Transition	Integer Value	2 to 512, default value of 10
Transition type	String	Rising or Falling
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Level	Voltage in volts	Can be user-specified when Specify checkbox is selected, or ignored if the checkbox is cleared.

Name	Type	Range
Specify	Boolean value	Selected or cleared

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

High Voltage Reference Level: VHRef

Summary

VHRef calculates the high (logical "1") voltage reference level. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators. This measurement is intended to be used for the analysis of digital circuits.

The Start and End Time are specified in time units explicitly.

This measurement uses the histogram method to determine the high reference level as follows. A histogram with 100 bins uses the absolute minimum voltage Vmin as the lowest voltage, and the absolute maximum voltage Vmax as the highest voltage. The middle of the most populated histogram bin located in the interval [Vmin, Vmin+0.4(Vmax-Vmin)] is reported as the low voltage reference level VLRef. The middle of the most populated histogram located in the interval [Vmin+0.6(Vmax-Vmin), Vmax] is reported as VHRef.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the high and low voltage reference levels on a sweep variable. Document frequency is one of the possible sweep variables.

Differential High Voltage Reference Level: VHRefD

Summary

VHRefD calculates the high (logical "1") voltage reference level. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators. This measurement is intended to be used for the analysis of digital circuits.

The Start and End Time are specified in time units explicitly.

This measurement uses the histogram method to determine the high reference level as follows. A histogram with 100 bins uses the absolute minimum voltage Vmin as the lowest voltage, and the absolute maximum voltage Vmax as the highest voltage. The middle of the most populated histogram bin located in the interval [Vmin, Vmin+0.4(Vmax-Vmin)] is reported as the low voltage reference level VLRef. The middle of the most populated histogram located in the interval [Vmin+0.6(Vmax-Vmin), Vmax] is reported as VHRef.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the high and low voltage reference levels on a sweep variable. Document frequency is one of the possible sweep variables.

Low Voltage Reference Level: VLRef

Summary

VLRef calculates the low (logical "0") voltage reference level. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators. This measurement is intended to be used for the analysis of digital circuits.

The Start and End Time are specified in time units explicitly.

VLRef uses the histogram method to determine the low reference level as follows. A histogram with 100 bins uses the absolute minimum voltage Vmin as the lowest voltage, and the absolute maximum voltage Vmax as the highest voltage. The middle of the most populated histogram bin located in the interval [Vmin, Vmin+0.4(Vmax-Vmin)] is reported as VLRef. The middle of the most populated histogram located in the interval [Vmin+0.6(Vmax-Vmin), Vmax] is reported as the high voltage reference level VHRef.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the high and low voltage reference levels on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Low Voltage Reference Level: VLRefD

Summary

VLRefD calculates the low (logical "0") voltage reference level. It can be used with Harmonic Balance, HSPICE transient, or Spectre transient simulators. This measurement is intended to be used for the analysis of digital circuits.

The Start and End Time are specified in time units explicitly.

VLRefD uses the histogram method to determine the low reference level as follows. A histogram with 100 bins uses the absolute minimum voltage Vmin as the lowest voltage, and the absolute maximum voltage Vmax as the highest voltage. The middle of the most populated histogram bin located in the interval [Vmin, Vmin+0.4(Vmax-Vmin)] is reported as VLRef. The middle of the most populated histogram located in the interval [Vmin+0.6(Vmax-Vmin), Vmax] is reported as the high voltage reference level VHRef.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Start Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Time in s, ms, us, ns, ps	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in voltage units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the high and low voltage reference levels on a sweep variable. Document frequency is one of the possible sweep variables.

Peak Voltage: VPeak

Summary

VPeak calculates the peak value of the voltage. This value, depending on the setting for Peak Type parameter can be either maximum or minimum. The user specifies the range of time values [Start time, End Time] to look for the peak. If no peak is found in the specified range, a warning message is issued to this effect.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real values	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Peak Type	String	Max or Min
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Differential Peak Voltage: VPeakD

Summary

VPeakD calculates the peak value of the voltage. This value, depending on the setting for Peak Type parameter can be either maximum or minimum. The user specifies the range of time values [Start time, End Time] to look for the peak. If no peak is found in the specified range, a warning message is issued to this effect.

Parameters

Name	Type	Range
Data Source Name	Schematic	0 to 1000 ports
+Measurement Component	Port or voltage probe	N/A
- Measurement Component	Port or voltage probe	N/A
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
End Time	Real values	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field
Peak Type	String	Max or Min
Data Source Name	Schematic	0 to 1000 ports
Measurement Component	Port or voltage probe	N/A
Start Time	Real Value	Positive value specified in seconds, microseconds, nanoseconds, or picoseconds depending on the value of the corresponding Units field

Result

This measurement returns a real value in time units.

Graph Type

This measurement can be displayed on a rectangular graph or table. It can be used to display the dependence of the rise or fall time on a sweep variable. Document frequency is one of the possible sweep variables.

Single-Tone Available Voltage Gain (Swept Power): AVG

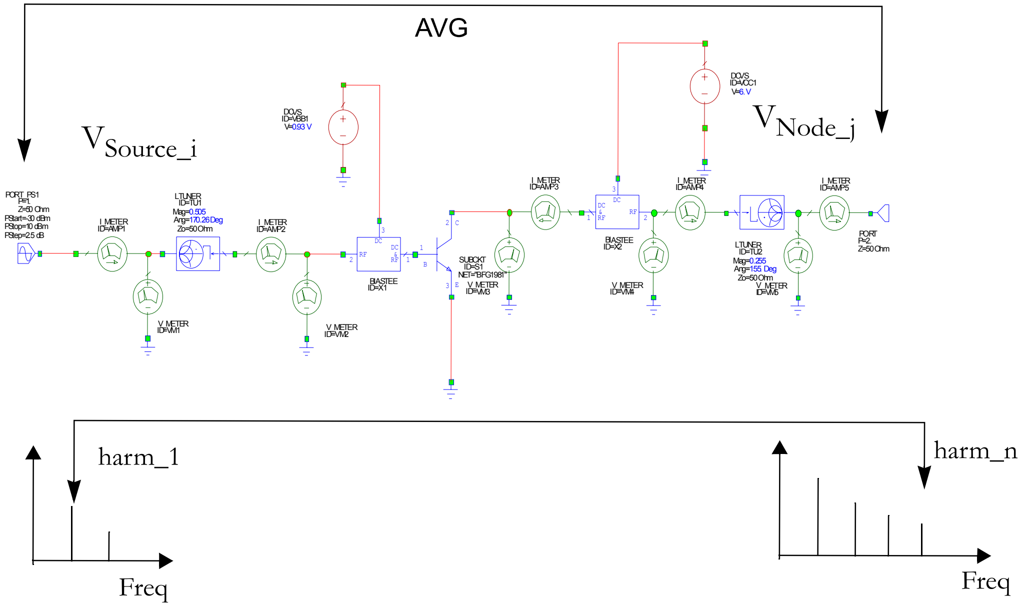
Summary

AVG computes the nonlinear voltage gain from the signal source to an arbitrary node's harmonic component in the RF circuit (RF Port or VMeter). The general equation for this implementation is:

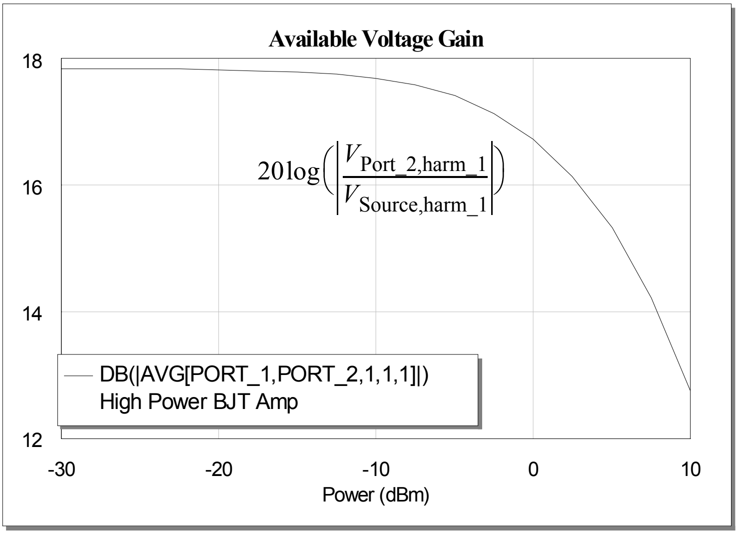
$$V_{\text{Source\_i,harm\_n}} = \sqrt{(PA_{\text{Source\_i,harm\_n}} \cdot 8 \cdot \Re(Z_{\text{Source\_i,harn}}}$$

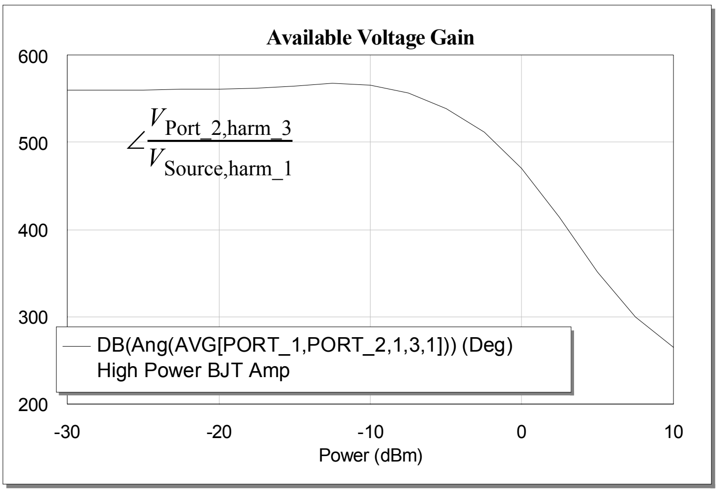
$$AVG_{\text{Node\_i,j,harm\_n,m}} = \frac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Source\_i,harm\_n}}}$$

where $PA_{\text{Source\_i,harm\_n}}$ indicates the available power at the node i and harmonic n. The following is an example of a typical RF single-stage amplifier topology utilizing biasing and matching blocks.



The available voltage gain between $\frac{V_{\text{Port\_2,harm\_1}}}{V_{\text{Source,harm\_1}}}$ and $\frac{V_{\text{Port\_2,harm\_3}}}{V_{\text{Source,harm\_1}}}$ is illustrated as follows:





Parameters

Name	Type	Range
Data Source Name	Subcircuit	0 to 1000
Voltage In Component	String	N/A
Voltage Out Component	String	N/A
Harmonic Indexes (Voltage In)	String	N/A
Harmonic Indexes (Voltage Out)	String	N/A
Frequency Sweep Index	Scalar	0 to EOL

Result

This measurement returns a complex value of nonlinear available voltage gain as a function of input power and harmonic components at a specified frequency.

Graph Type

This measurement displays in Rectangular or Polar format. You can display its magnitude in dB by selecting **DB** as the **Result Type** in the Add/Modify Measurement dialog box. The x-axis for this measurement is in power sweep.

Relationship to Transducer Voltage Gain

Following equation expresses the relation between Available voltage gain and Transducer voltage gain.

$$\begin{aligned} AVG_{\text{Node\_i,j,harm\_n,m}} &= \frac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Source\_i,harm\_n}}} = \frac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Node\_i,harm\_n}}} \cdot \frac{V_{\text{Node\_i,harm}}}{V_{\text{Source\_i,harm}}} \\ &= TVG_{\text{Node\_i,j,harm\_n,m}} \cdot \frac{V_{\text{Node\_i,harm\_n}}}{V_{\text{Source\_i,harm\_n}}} \\ &= TVG_{\text{Node\_i,j,harm\_n,m}} \cdot \frac{Z_{\text{Node\_i,harm\_n}}}{Z_{\text{Node\_i,harm\_n}} + Z_{\text{Source\_i}}} \end{aligned}$$

From above equation we can infer that:

$$\begin{aligned} 20\log(|AVG_{\text{Node\_i,j,harm\_n,m}}|) &= 20\log(|TVG_{\text{Node\_i,j,harm\_n,m}}|) \\ &+ 20\log\left(\left|\frac{Z_{\text{Node\_i,harm\_n}}}{Z_{\text{Node\_i,harm\_n}} + Z_{\text{Source\_i,h}}}\right.\right. \end{aligned}$$

In a special case when $Z_{\text{Node\_1,harm\_1}} = Z_{\text{Source,harm\_1}} = \alpha$, where α is a constant real value, e.g. 50 Ω, the above equation can be rewritten as

$$20\log(|AVG_{Node\_i,j,harm\_n,m}|) = 20\log(|TVG_{Node\_i,j,harm\_n,m}|) -$$

Relationship to Output Power Level

Following equation expresses the relation between Available voltage gain and power level of an arbitrary node as;

$$P_{Node\_j,harm\_m} = |AVG_{Node\_i,j,harm\_n,m}|^2 \cdot PA_{Source\_i,harm\_n} \cdot \Re(Z_{Source\_i,harm\_n}) \cdot \Re\left(\frac{1}{Z_{Node\_i,harm\_n}}\right) \cdot 4$$

or

$$10\log(P_{Node\_j,harm\_m}) = 20\log(|AVG_{Node\_i,j,harm\_n,m}|) + 10\log(PA_{Source\_i} + 10\log\left(\Re(Z_{Source\_i,harm\_n}) \cdot \Re\left(\frac{1}{Z_{Node\_i,harm\_n}}\right.\right.$$

In a special case when $Z_{Node\_1,harm\_1} = Z_{Source,harm\_1} = \alpha$,

where α is a constant real value, e.g. 50 Ω, the above equation can be rewritten as:

$$10\log(P_{Node\_j,harm\_m}) = 20\log(|AVG_{Node\_i,j,harm\_n,m}|) + 10\log(PA_{Source\_i,harm\_}$$

Left-Hand Circular Polarization (Sweep Phi): CE_LHCP

Summary

CE_LHCP is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the CON_LHCP measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See CON_LHCP for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Theta (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

E-Phi Pattern (Sweep Phi): CE_Phi

Summary

CE_Phi is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the CON_EPHI measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See CON_EPHI for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Right-Hand Circular Polarization (Sweep Phi): CE_RHCP

Summary

CE_RHCP is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the CON_RHCP measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See CON_RHCP for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to 1000 ports
Phi (theta)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

E-Theta Pattern (Sweep Phi): CE_Theta

Summary

CE_Theta is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the CON_ETheta measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See CON_ETheta for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Total Radiated Power (Sweep Phi): CP_Rad

Summary

CP_Rad is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the CON_TPwr measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See CON_TPwr for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Theta (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Left-Hand Circular Polarization (Sweep Theta): E_LHCP

Summary

E_LHCP is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the PPC_LHCP measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See PPC_LHCP for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Phi (degrees)	Real value	0 to 180
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

E-Phi Pattern (Sweep Theta): E_Phi

Summary

E_Phi is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the PPC_EPhi measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See PPC_EPhi for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Phi (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Right-Hand Circular Polarization (Sweep Theta): E_RHCP

Summary

E_RHCP is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the PPC_RHCP measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See PPC_RHCP for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Phi (degrees)	Real value	0 to 180
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

E-Theta Pattern (Sweep Theta): E_Theta

Summary

E_Theta is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the PPC_ETheta measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See PPC_ETheta for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Phi (degrees)	Real value	-90 to 90
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Total Radiated Power (Sweep Theta): P_Rad

Summary

P_Rad is retained to allow compatibility with pre-existing projects. All future measurements of this type should use the PPC_TPwr measurement type. Importantly, this measurement has been modified to normalize this result to the total power radiated (See PPC_TPwr for details).

Parameters

Name	Type	Range
EM Structure Name	Subcircuit	1 to1000 ports
Phi (degrees)	Real value	0 to 180
Frequency Sweep Index	Integer value	1 to 1000

Result

This measurement returns a real value. The x-axis for this measurement is in angle units.

Graph Type

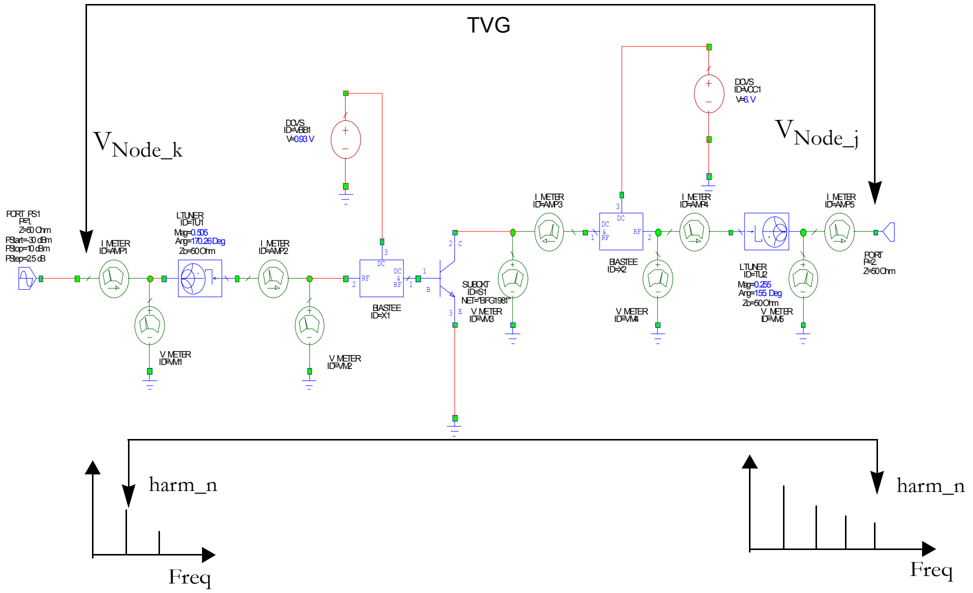
This measurement can be displayed on a an antenna plot, rectangular graph or tabular grid.

Single-Tone Transducer Voltage Gain (Swept Power): TVG

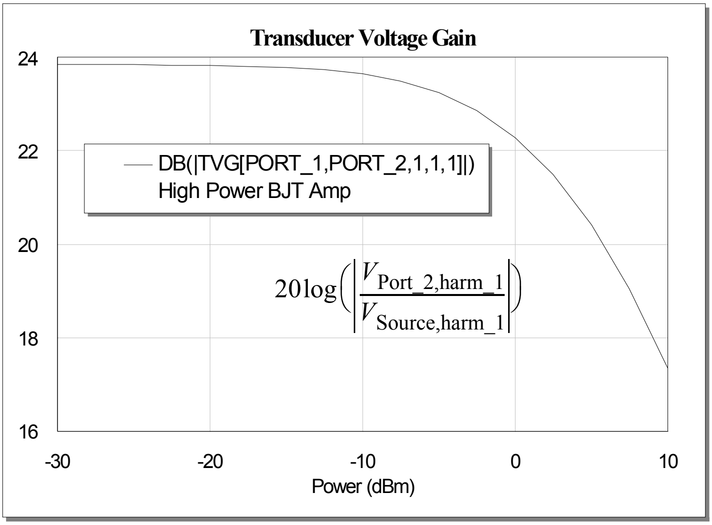
Summary

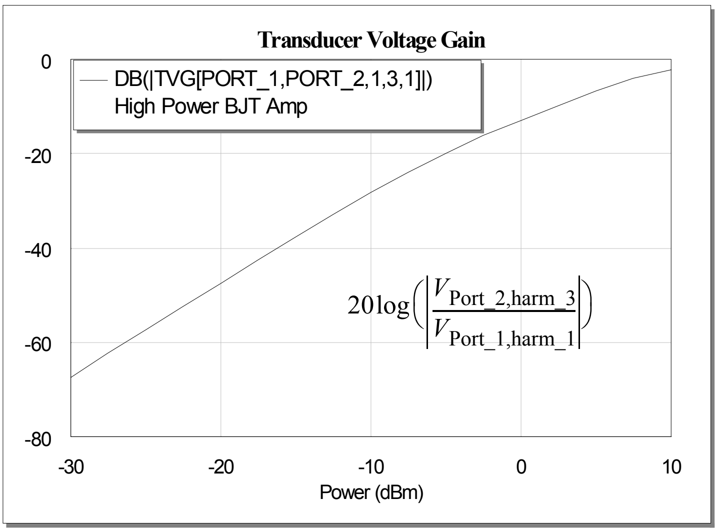
TVG computes the nonlinear transducer voltage gain between two arbitrary nodes' harmonics in the RF circuit. The general equation for this implementation is: $TVG_{\text{Node\_i,j,harm\_n,m}} = \dfrac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Node\_i,harm\_n}}}$

The following is an example of a typical RF single-stage amplifier topology utilizing biasing and matching blocks.



The transducer voltage gain between $\dfrac{V_{\text{Port\_2,harm\_1}}}{V_{\text{Port\_1,harm\_1}}}$ and $\dfrac{V_{\text{Port\_2,harm\_3}}}{V_{\text{Port\_1,harm\_1}}}$ are illustrated as follows:


Transducer Voltage Gain
DB(|TVG[PORT_1,PORT_2,1,1,1]|)
High Power BJT Amp
$20\log\left(\left|\frac{V_{Port\_2,harm\_1}}{V_{Source,harm\_1}}\right|\right)$
24
22
20
18
16
-30
-20
-10
0
10
Power (dBm)


Transducer Voltage Gain
DB(|TVG[PORT_1,PORT_2,1,3,1]|)
High Power BJT Amp
$20\log\left(\left|\frac{V_{Port\_2,harm\_3}}{V_{Port\_1,harm\_1}}\right|\right)$
0
-20
-40
-60
-80
-30
-20
-10
0
10
Power (dBm)

Parameters

Name	Type	Range
Data Source Name	Subcircuit	0 to 1000
Voltage In Component	String	N/A
Voltage Out Component	String	N/A
Harmonic Indexes (Voltage In)	String	N/A
Harmonic Indexes (Voltage Out)	String	N/A
Frequency Sweep Index	Scalar	0 to EOL

Result

This measurement returns a complex value of nonlinear transducer voltage gain as a function of input power and harmonic components at a specified frequency.

Graph Type

This measurement displays in Rectangular or Polar format. You can display its magnitude in dB by selecting **DB** as the **Result Type** in the Add/Modify Measurement dialog box. The x-axis for this measurement is in power sweep.

Relationship to Available Voltage Gain

Following equation expresses the relation between Available voltage gain and Transducer voltage gain.

$$
\begin{aligned}
TVG_{\text{Node\_i,j,harm\_n,m}} &= \frac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Node\_i,harm\_n}}} = \frac{V_{\text{Node\_j,harm\_m}}}{V_{\text{Source\_i,harm\_n}}} \cdot \frac{V_{\text{Source\_i,harm\_n}}}{V_{\text{Node\_i,harm\_n}}} \\
&= AVG_{\text{Node\_i,j,harm\_n,m}} \cdot \frac{V_{\text{Source\_i,harm\_n}}}{V_{\text{Node\_i,harm\_n}}} \\
&= AVG_{\text{Node\_i,j,harm\_n,m}} \cdot \left(\frac{Z_{\text{Node\_i,harm\_n}}}{Z_{\text{Node\_i,harm\_n}} + Z_{\text{Source\_i,harm\_n}}}\right)^{-1}
\end{aligned}
$$

From above equation we can infer that:

$$
\begin{aligned}
20\log(|TVG_{\text{Node\_i,j,harm\_n,m}}|) &= 20\log(|AVG_{\text{Node\_i,j,harm\_n,m}}|) \\
&\quad - 20\log\left(\left|\frac{Z_{\text{Node\_i,harm\_n}}}{Z_{\text{Node\_i,harm\_n}} + Z_{\text{Source\_i,harm\_n}}}\right|\right)
\end{aligned}
$$

In a special case when $Z_{Node\_1,harm\_1} = Z_{Source,harm\_1} = \alpha$,

where α is a constant real value, e.g. 50 Ω, the above equation can be rewritten as

$$20\log(|TVG_{Node\_i,j,harm\_n,m}|) = 20\log(|AVG_{Node\_i,j,harm\_n,m}|) + 6\text{dB}$$

Relationship to Output Power Level

Following equation expresses the relation between Available voltage gain and power level of an arbitrary node as;

$$P_{Node\_j,harm\_m} = |TVG_{Node\_i,j,harm\_n,m}|^2 \cdot P_{Node\_i,harm\_n}$$

or

$$10\log(P_{Node\_j,harm\_m}) = 20\log(|TVG_{Node\_i,j,harm\_n,m}|) + 10\log(P_{Node\_i,harm\_n})$$

Voltage Spectrum: VSD

Summary

VSD is replaced by the Visual System Simulator (VSS) System Spectrum V_SPEC measurement.

Normalized Voltage Spectrum: VSDN

Summary

VSDN is replaced by the Visual System Simulator (VSS) System Spectrum V_SPECN measurement.

Select Output Equation: Eqn

Summary

Output equations are the result of user controlled data manipulation. The source of the data is typically simulation results and then mathematical operations can be performed on the data. There many uses for output equations. See the “Using Output Equations” section of the “Variables and Equations” chapter in the *Microwave Office/Analog Office User Guide* for details on yield simulation.

Parameters

Name	Type	Range
Equation Name	String	N/A

Result

Eqn 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 real value can be displayed in dB by selecting the **DB** check box.

Component Sensitivity Histogram: YSens

Summary

YSens is used for plotting component sensitivity histograms. See the “Yield Analysis” section of the “Optimizing, Tuning, and Yield” chapter in the *Microwave Office/Analog Office User Guide* for details on yield simulation.

Parameters

Name	Type	Range
Variables In	Subcircuit	0 to 100 ports
Circuit Component	String	N/A

NOTE. All measurements will have additional parameters that allow you to specify the plotting configuration for swept parameters. These parameters are dynamic; they change based upon which data source is selected. Please see the “Swept Parameter Analysis” chapter in the *Microwave Office/Analog Office User Guide* for details on configuring these parameters. Use caution when running yield analysis on variables that are also being swept. Each yield iteration will perform the swept analysis and could give you answers you don’t expect.

Result

This measurement returns a real value.

INDEX

F

G

H

I

J

K

L

M

N