CosmosScope™
Reference Manual

Version W-2004.12, December 2004

SYNOPSYS®

Copyright Notice and Proprietary Information

Right to Copy Documentation

The license agreement with Synopsys permits licensee to make copies of the documentation for its internal use only. Each copy shall include all copyrights, trademarks, service marks, and proprietary rights notices, if any. Licensee must assign sequential numbers to all copies. These copies shall contain the following legend on the cover page:

> "This document is duplicated with the permission of Synopsys, Inc., for the exclusive use of _____ and its employees. This is copy number _____."

Destination Control Statement

All technical data contained in this publication is subject to the export control laws of the United States of America. Disclosure to nationals of other countries contrary to United States law is prohibited. It is the reader's responsibility to determine the applicable regulations and to comply with them.

Disclaimer

SYNOPSYS, INC., AND ITS LICENSORS MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

Registered Trademarks (®)

Synopsys, AMPS, Arcadia, C Level Design, C2HDL, C2V, C2VHDL, Cadabra, Calaveras Algorithm, CATS, CSim, Design Compiler, DesignPower, DesignWare, EPIC, Formality, HSPICE, Hypermodel, I, iN-Phase, in-Sync, Leda, MAST, Meta, Meta-Software, ModelAccess, ModelTools, NanoSim, OpenVera, PathMill, Photolynx, Physical Compiler, PowerMill, PrimeTime, RailMill, Raphael, RapidScript, Saber, SiVL, SNUG, SolvNet, Stream Driven Simulator, Superlog, System Compiler, Testify, TetraMAX, TimeMill, TMA, VCS, Vera, and Virtual Stepper are registered trademarks of Synopsys, Inc.

Trademarks (™)

abraCAD, abraMAP, Active Parasitics, AFGen, Apollo, Apollo II, Apollo-DPII, Apollo-GA, ApolloGAII, Astro, Astro-Rail, Astro-Xtalk, Aurora, AvanTestchip, AvanWaves, BCView, Behavioral Compiler, BOA, BRT, Cedar, ChipPlanner, Circuit Analysis, Columbia, Columbia-CE, Comet 3D, Cosmos, CosmosEnterprise, CosmosLE, CosmosScope, CosmosSE, Cyclelink, Davinci, DC Expert, DC Expert *Plus*, DC Professional, DC Ultra, DC Ultra Plus, Design Advisor, Design Analyzer, Design Vision, DesignerHDL, DesignTime, DFM-Workbench, DFT Compiler, Direct RTL, Direct Silicon Access, Discovery, DW8051, DWPCI, Dynamic-Macromodeling, Dynamic Model Switcher, ECL Compiler, ECO Compiler, EDAnavigator, Encore, Encore PQ, Evaccess, ExpressModel, Floorplan Manager, Formal Model Checker, FoundryModel, FPGA Compiler II, FPGA *Express*, Frame Compiler, Galaxy, Gatran, HDL Advisor, HDL Compiler, Hercules, Hercules-Explorer, Hercules-II, Hierarchical Optimization Technology, High Performance Option, HotPlace, HSPICE-Link, iN-Tandem, Integrator, Interactive Waveform Viewer, i-Virtual Stepper, Jupiter, Jupiter-DP, JupiterXT, JupiterXT-ASIC, JVXtreme, Liberty, Libra-Passport, Library Compiler, Libra-Visa, Magellan, Mars, Mars-Rail, Mars-Xtalk, Medici, Metacapture, Metacircuit, Metamanager, Metamixsim, Milkyway, ModelSource, Module Compiler, MS-3200, MS-3400, Nova Product Family, Nova-ExploreRTL, Nova-Trans, Nova-VeriLint, Nova-VHDLlint, Optimum Silicon, Orion_ec, Parasitic View, Passport, Planet, Planet-PL, Planet-RTL, Polaris, Polaris-CBS, Polaris-MT, Power Compiler, PowerCODE, PowerGate, ProFPGA, ProGen, Prospector, Protocol Compiler, PSMGen, Raphael-NES, RoadRunner, RTL Analyzer, Saturn, ScanBand, Schematic Compiler, Scirocco, Scirocco-i, Shadow Debugger, Silicon Blueprint, Silicon Early Access, SinglePass-SoC, Smart Extraction, SmartLicense, SmartModel Library, Softwire, Source-Level Design, Star, Star-DC, Star-MS, Star-MTB, Star-Power, Star-Rail, Star-RC, Star-RCXT, Star-Sim, Star-SimXT, Star-Time, Star-XP, SWIFT, Taurus, Taurus-Device, Taurus-Layout, Taurus-Lithography, Taurus-Process, Taurus-Topography, Taurus-Visual, Taurus-Workbench, TimeSlice, TimeTracker, Timing Annotator, TopoPlace, TopoRoute, Trace-On-Demand, True-Hspice, TSUPREM-4, TymeWare, VCS Express, VCSi, Venus, Verification Portal, VFormal, VHDL Compiler, VHDL System Simulator, VirSim, and VMC are trademarks of Synopsys, Inc.

Service Marks (^{SM})

MAP-in, SVP Café, and TAP-in are service marks of Synopsys, Inc.

SystemC is a trademark of the Open SystemC Initiative and is used under license.
ARM and AMBA are registered trademarks of ARM Limited.
All other product or company names may be trademarks of their respective owners.

Printed in the U.S.A.

Document Order Number: 00000-000 WA
Family Name Product Name Manual Type, version W-2004.12

Table Of Contents

Chapter 1

Introduction

CosmosScope is a graphical waveform analyzer tool that allows you to view and analyze simulation results in the form of waveforms displayed on graphs, or as values displayed in lists. Tools available with CosmosScope include:

- the Signal Manager, through which the plotfiles are opened, filtered, and placed into a graph window or calculator
- the Measurement Tool, which provides over 50 measurements that can be applied to a waveform
- the Waveform Calculator, which emulates a hand-held calculator that interacts graphically with the application
- the Command Line Tool, which allows you to enter AIM commands, write scripts, and save them into files

Invoking CosmosScope

There are a number of different ways to invoke Scope, including:

- On a Windows system, use the command line invocation described below, or simply select:

  Programs > *{install_location}* **> Scope**

  or

  Programs > *{install_location}* **> CosmosScope**

- To invoke the product on UNIX systems, see the command line invocation instructions, below.

Command Line Invocation and Options

CosmosScope can be executed from the UNIX command prompt or from the Command Prompt window. The full form of the `scope` and `cscope` commands for CosmosScope is shown below:

`cscope` *[-h][*-display *host[*`:`*server.display]]*
[-pfiles *pfilename][pfile...]][*-noconfig*][*-geom *geom]*
[-script *aimfile*

The following table describes the `scope` and `cscope` command options.

Option	Description
`-h`	Displays the `scope` (or `cscope`) command syntax and a list of the invocation options.
`-display` *host*`:0.0`	Displays screen graphics on the specified host. On some systems, you can replace *host*`:0.0` with `unix:0.0` or `:0.0`, when the display host is the one running the simulator (or the Scope Waveform Analyzer).
`-pfiles` *pfile*	Specifies the plotfile to be opened at start-up.
`-noconfig`	Requests that the saved configuration not be loaded on start-up.
`-geom` *geom*	Defines the geometry for the Scope window.
`-script` *aimfile*	Executes the specified AIM script on start-up.

Option	Description
`-app` *application name*	Specifies the application that CosmosScope is integrated with. The value can be `saber`, `cosmos`, or `saberhdl`.

Opening a Plot File

To open a plot file:

- Choose the **File > Open... > Plotfile...** menu choice. This choice displays the Open Plot Files dialog box.
- In the Directory field, navigate to the directory that contains the plot file you wish to analyze.
- Set the Files of type field as appropriate for the kind of plot file you wish to open.
- Highlight the desired file and click the **Open** button. Refer to the information on the Signal Manager tool to begin your analysis.

Tutorials

The following topics provide tutorials on how to use CosmosScope to view different waveforms:

- Tutorial: Viewing Saber Simulator Results
- Tutorial: Viewing HSPICE Results

CosmosScope also reads AWD, FSDB Version 2.3 (EPIC, VERILOG), VCD, TouchStone, Star-SimXT and Polaris plot files. While these output formats are not covered in these tutorials, the process of opening these files and using CosmosScope with them is similar to the process shown in these tutorials.

Chapter 2

Tutorial: Viewing Saber Simulator Results

In this tutorial you will use CosmosScope to view analysis results from the simulation of a single-stage amplifier design.

This tutorial is divided into the following topics:

- Setting up the Saber Simulator Data
- Viewing Saber Transient Analysis Waveforms
- Viewing Saber AC Analysis Waveforms
- Performing Measurements on a Waveform

Setting up the Saber Simulator Data

Saber Simulator analysis results for a simple transistor amplifier have been provided for use with this tutorial. Create a directory and make a copy of the example as follows:

1. Create a directory called `synopsys_tutorial`.
2. Navigate to the new `synopsys_tutorial` directory.
3. Copy the *install_home*`/examples/Saber/SaberScope/saber_amp` directory to the `synopsys_tutorial` directory:

   UNIX:

   cp -r *install_home*/examples/Saber/SaberScope/saber_amp .

   install_home is the location where your software has been installed.

   Windows:

   In Windows Explorer, hold down the `Ctrl` key and drag the `saber_amp` folder from `\examples\Saber\SaberScope\` to the `synopsys_tutorial` directory you just created.

Viewing Saber Transient Analysis Waveforms

The results of a Saber Simulator transient analysis reside in the `saber_amp` directory. You can view the results with the CosmosScope Waveform Analyzer as follows:

1. Invoke CosmosScope.
2. Open the Open Plotfiles dialog box: **File > Open > Plotfiles**.
3. In the Open Plotfiles dialog box, browse to the `synopsys_tutorial\saber_amp` directory; in the Files of type field, select `Saber pl (*.ai_pl, *.p1, *.p1*)`.
4. Click on the `single_amp.tr.ai_pl` item and click the **Open** button. The Signal Manager and the single_amp.tr.ai_pl Plot File windows are displayed.
5. From the single_amp.tr.ai_pl Plot File window, select signal `in` by left-clicking it. The signal is highlighted.
6. Plot the selected signal on the graph by clicking the **Plot** button.
7. In the single_amp.tr.ai_pl Plot File window, select signal `aout`.
8. Plot the selected signal on the same graph as the `in` signal by moving the cursor to the Graph window and clicking the middle mouse button. When using a two-button mouse, place the cursor in the graph region, click the right mouse button to bring up the graph pop-up, then select **Plot**.

   These waveforms show the input and the output of a simple transistor amplifier.
9. Zoom in to the area between 2u and 4u by moving the cursor to the X-axis 2u tick mark.
10. Click-and-hold the left mouse button and drag it over to the 4u tick mark and release the button. The same technique can be used to zoom on the Y-axis.
11. If you like, experiment with the Zoom icons  .
12. When you have finished viewing the waveforms, click the Clear icon  .

Viewing Saber AC Analysis Waveforms

The results of a Saber Simulator AC analysis also reside in the `saber_amp` directory. You can view these results as follows:

1. In the Signal Manager dialog box, click on the **Open Plotfiles** button.
2. In the Open Plotfiles dialog box, click on the `single_amp.ac.ai_pl` selection and click the **Open** button. The single_amp.ac.ai_pl Plot File window is displayed.
3. In the single_amp.ac.ai_pl Plot File window, select signal `aout` and plot it.
4. In this tutorial you do not need the `Phase(deg):f(Hz)` waveform. To delete it from the Graph window, do the following:
   a. Move the mouse cursor to the `aout` signal name associated with the `Phase(deg):f(Hz)` plot. The `aout` signal name and the waveform change color.
   b. Press-and-hold the right mouse button to bring up the Signal Menu.
   c. Select the **Delete Signal** item.
5. Do the following to see how you can plot additional waveforms to the Graph window:
   a. From the single_amp.tr.ai_pl Plot File window, plot the `aout` and `in` signals. Two new waveforms are added to the graph window.
   b. Delete the `in` and `aout` waveforms when you have finished viewing them.
6. Look at the `aout dB(V):f(Hz)` (dB in volts versus frequency in Hertz) waveform in the Graph window.

   From the waveform you can see that the gain is about 10dB from about 2000 Hz to 300 kHz. The next part of this tutorial uses the Measurement Tool on this waveform to get some accurate readings on the gain and the frequency response.

Performing Measurements on a Waveform

The Measurement Tool allows you to perform various measurements on a waveform. Check the bandwidth and gain of the single-stage amplifier output signal (aout) as follows:

1. Use the **Close** buttons on the Plot File windows and the Signal Manager dialog box to close them.
2. In the Tool Bar located at the bottom of the CosmosScope window, click the Measurement icon  .

   The Measurement dialog box appears.
3. Select the Bandwidth measurement in the Measurement dialog box as follows:

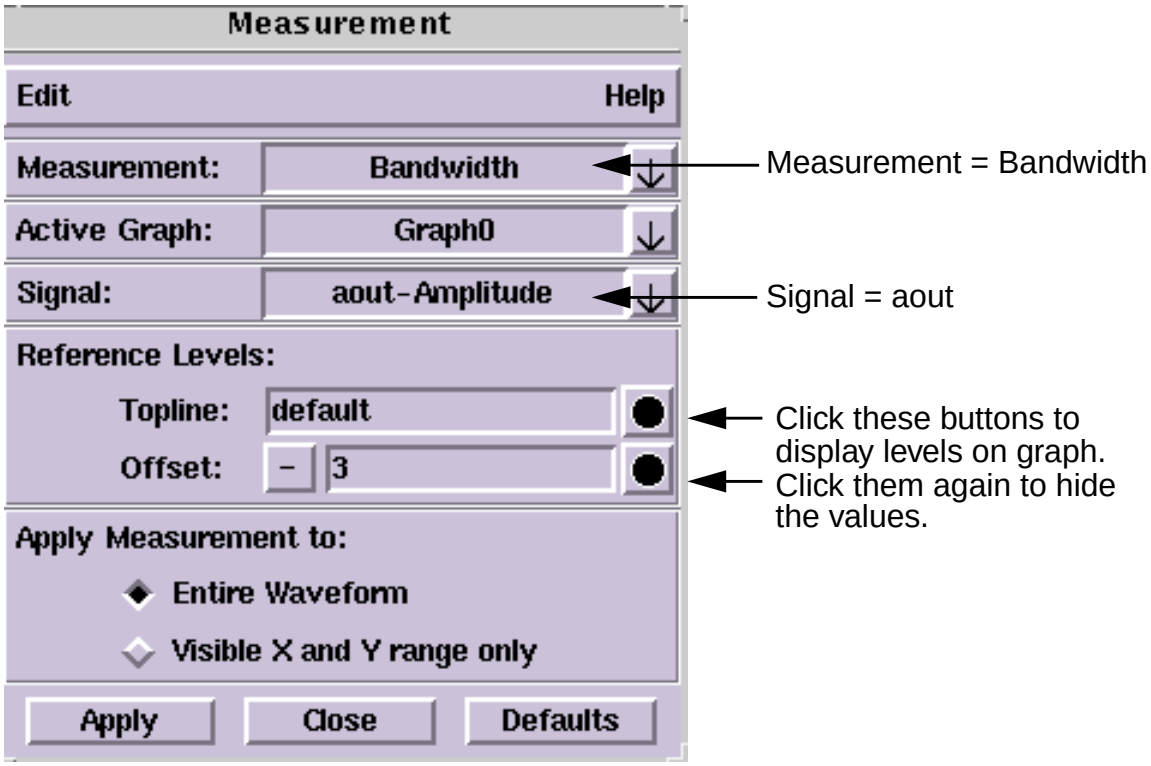
   a. Move the mouse cursor to the right of the Measurement field and press and hold the left mouse button on the down arrow button.

   b. Move the mouse cursor down to the **Frequency Domain** menu.

   c. Select **Bandwidth**.

   To summarize, choose the **Measurement > Frequency Domain > Bandwidth** menu item.

d. Because there is only one signal in the Graph window, `aout` should appear in the Signal field in the Measurement dialog box as shown in the following figure.



e. If you want to see values displayed on the graph for Topline and Offset that are used in the bandwidth calculation, click the visibility indicator buttons to the right of the Reference Levels fields they will turn green to indicate they're activated.

f. Click the **Apply** button. The bandwidth is displayed on the graph.

4. Select the Gain Margin measurement by doing the following:

   a. Choose the **Measurement > Frequency Domain > Gain Margin** menu item.

   b. Click the **Apply** button. The gain margin is displayed on the graph.

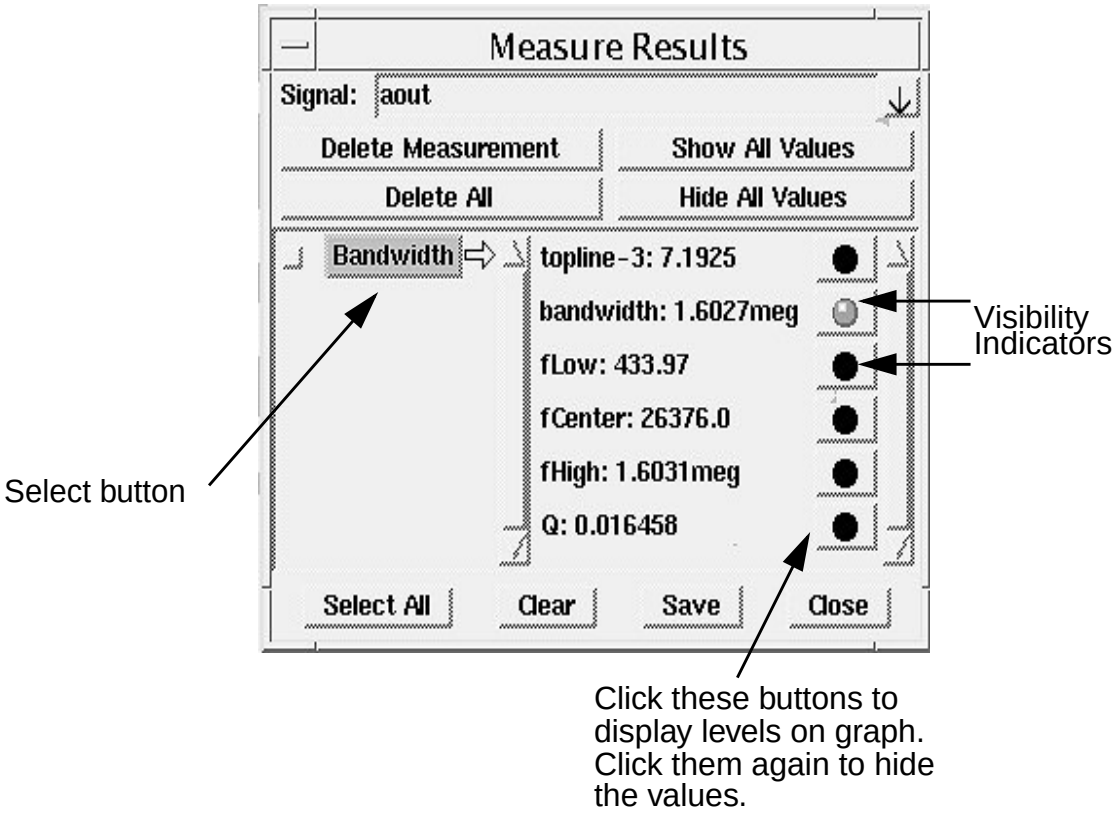
   c. You can select the measurement labels and move them if the graph becomes too cluttered. Position the cursor over the text. Then left-click and hold while moving the cursor to a new location.

5. You can get more information about each of the measures you performed or control the amount of information displayed in the Graph window by using the Measure Results dialog box as follows:

a. In the Graph window, move the mouse cursor to the `aout` signal name.

b. Use the popup menu and choose the **Signal Menu > Measure Results...** item. A Measure Results dialog box appears.

c. In the Measure Results dialog box, be sure the Bandwidth item in the left column is selected as shown in the following figure:



d. Notice in the Measure Results dialog box, in the right column, the different values that are available from executing the bandwidth measurement.

e. Click on the various visibility indicators to choose which values are displayed in the Graph window.

f. When you have finished exploring the Measure Results dialog box, close it.

6. To close CosmosScope, select the **File > Exit** menu item.

This concludes the tutorial for analyzing Saber Simulator results.

Chapter 3

Tutorial: Viewing HSPICE Results

In this tutorial you use CosmosScope to view the analysis results from the simulation of a single-stage amplifier design.

This tutorial is divided into the following topics:

- Setting up the Design Data
- Viewing HSPICE Transient Analysis Waveforms
- Viewing AC Analysis Waveforms
- Performing Measurements on a HSPICE Waveform

Setting up the Design Data

Analysis-results from a simple transistor amplifier have been created for you using the HSPICE transient and AC simulators for use with this tutorial. You will create a directory and then make a copy of the example as follows:

1. Create a directory called `synopsys_tutorial`.
2. Navigate to the new `synopsys_tutorial` directory.
3. Copy the *install_home*`/examples/Saber/CScope/hspice_amp` directory to the `synopsys_tutorial` directory:

   UNIX:

   `cp -r` *install_home*`/examples/Saber/CScope/hspice_amp`

   install_home is the location where your software has been installed.

   Windows:

   In Explorer, hold down the `Ctrl` key and drag the `hspice_amp` folder from *install_home*`\examples\Saber\CScope\` to the `synopsys_tutorial` directory that you just created.

Viewing HSPICE Transient Analysis Waveforms

The results of a simulator transient analysis reside in the `hspice_amp` directory. You can view the results with the CosmosScope Waveform Analyzer as follows:

1. Invoke CosmosScope.
2. Open the Open Plotfiles dialog box: **File > Open > Plotfiles**.
3. In the Open Plotfiles dialog box, browse to the `synopsys_tutorial\hspice_amp` directory; in the Files of type field, select `HSPICE (*.tr*, *.ac*, *.sw*, *.ft*)` item.
4. Click on the `amp.tr0` item and click the **Open** button. The Signal Manager and the amp Plot File windows are displayed.
5. From the amp Plot File window, select signal `v(in)` by left-clicking it. The signal is highlighted.
6. Plot the selected signal on the graph by clicking the **Plot** button.
7. In the amp Plot File window, select signal `v(aout)`.
8. Plot the selected signal on the same graph as the `v(in)` signal by moving the cursor to the Graph window and clicking the middle mouse button. When using a two-button mouse, place the cursor in the graph region, click the right mouse button to bring up the graph pop-up, then select **Plot**.

   These waveforms show the input and the output of a simple transistor amplifier.
9. Zoom in to the area between 2u and 4u by moving the cursor to the X-axis 2u tick mark.
10. Click-and-hold the left mouse button and drag it over to the 4u tick mark and release the button. The same technique can be used to zoom on the Y-axis.
11. If you like, experiment with the Zoom icons  .
12. When you have finished viewing the waveforms, click the Clear icon  .

Viewing AC Analysis Waveforms

The results of a simulator AC analysis also reside in the `hspice_amp` directory. You can view these results as follows:

1. In the Signal Manager dialog box, click on the **Open Plotfiles** button.
2. In the Open Plotfiles dialog box, in the Files of type field, select the `HSPICE (*.tr*, *.ac*, *.sw*, *.ft*)` item.
3. Click on the `a.ac0` selection and click the **Open** button. The "a" Plot File window is displayed.
4. In the Plot File window, select signal `v(aout)` and plot it.
5. In this tutorial you do not need the `Phase(deg):Frequency(Hertz)` waveform. To delete it from the Graph window, do the following:
   a. Move the mouse cursor to the `v(aout)` signal name associated with the Phase plot. The `v(aout)` signal name and the waveform change color.
   b. Press-and-hold the right mouse button to bring up the Signal Menu.
   c. Select the **Delete Signal** item.
6. Change the X-axis attributes to display as a logarithmic waveform as follows:
   a. To bring up the Axis Menu, move the cursor to the X-axis and click-and-hold the right mouse button.
   b. To bring up the Axis Attributes dialog box, select the **Attributes** menu item.
   c. In the Scale field, click the Log radio button. The waveform should now look similar to a bell curve.
   d. Close the Axis Attributes dialog box.
7. Do the following to see how you can plot additional waveforms to the Graph window:
   a. From the amp Plot File window, plot the `v(aout)` and `v(in)` signals. Two new waveforms are added to the graph window.
   b. When you have finished viewing the `v(in)` and `v(aout)` waveforms that you just plotted in the previous step, delete them as follows:

      First move the cursor to the waveform name on the graph. Then select the **Signal Menu > Delete Signal** menu item by right-clicking the mouse button. Do this for each signal you want to delete.

8. Look at the `vdb(aout) dB(V):Frequency(Hertz)` waveform in the Graph window.

   From the waveform you can see that the gain is about 10dB from about 2000 Hz to 300 kHz. The next part of this tutorial uses the Measurement Tool on this waveform to get some accurate readings on the gain and the frequency response.

Performing Measurements on an HSPICE Waveform

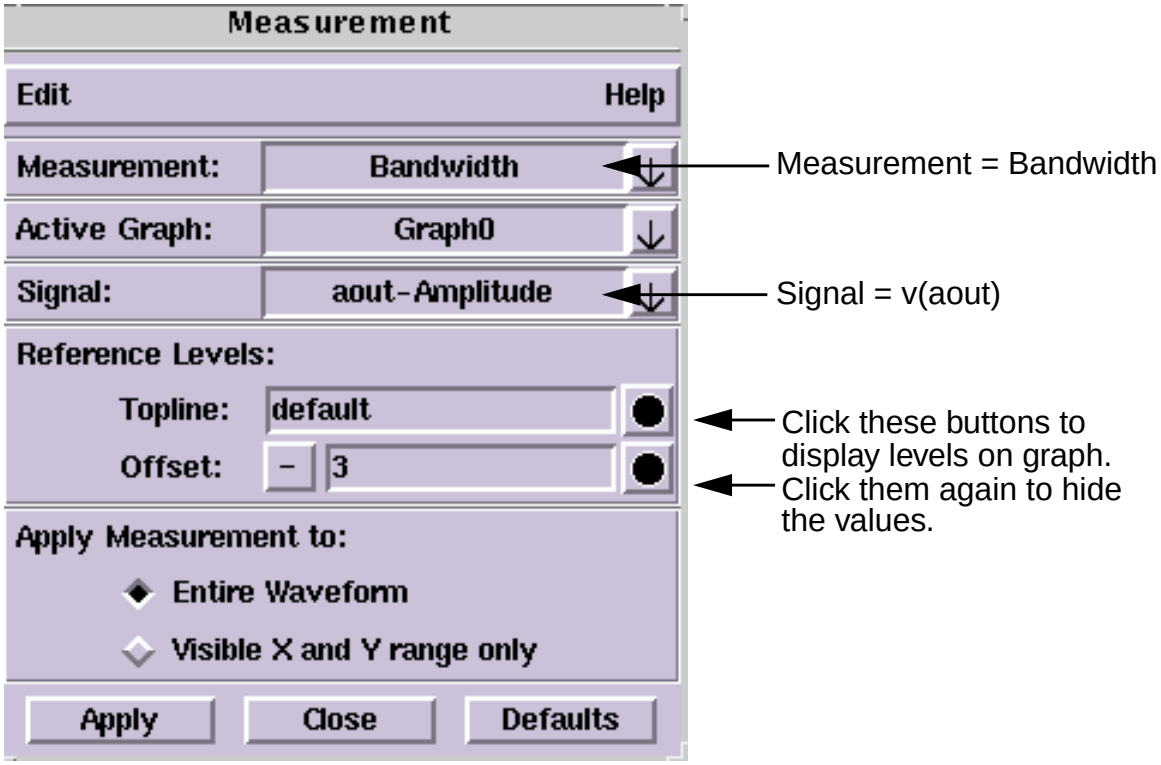
The Measurement Tool within CosmosScope provides a method of performing various measurements on a waveform. You check the bandwidth and gain of the single-stage amplifier output signal v(`aout`) as follows:

1. Close the Plot File windows and the Signal Manager window.
2. In the Tool Bar located at the bottom of the CosmosScope window, click the Measurement icon  .

   The Measurement dialog box appears.
3. Select the Bandwidth measurement in the Measurement dialog box as follows:
   a. Move the mouse cursor to the right of the Measurement field and press and hold the left mouse button on the down arrow button.
   b. Move the mouse cursor down to the **Frequency Domain** menu.
   c. Select **Bandwidth**.

      To summarize, choose the **Measurement > Frequency Domain > Bandwidth** menu item.

d. Because there is only one signal in the Graph window, v(aout) should appear in the Signal field in the Measurement dialog box as shown in the following figure.



e. If you want to see values displayed on the graph for Topline and Offset that are used in the bandwidth calculation, click the visibility indicator buttons to the right of the perspective Reference Levels fields.

f. Click the **Apply** button. The bandwidth is displayed on the graph.

4. Select the Gain Margin measurement by doing the following:

a. Choose the **Measurement > Frequency Domain > Gain Margin** menu item.

b. Click the **Apply** button. The gain margin is displayed on the graph.

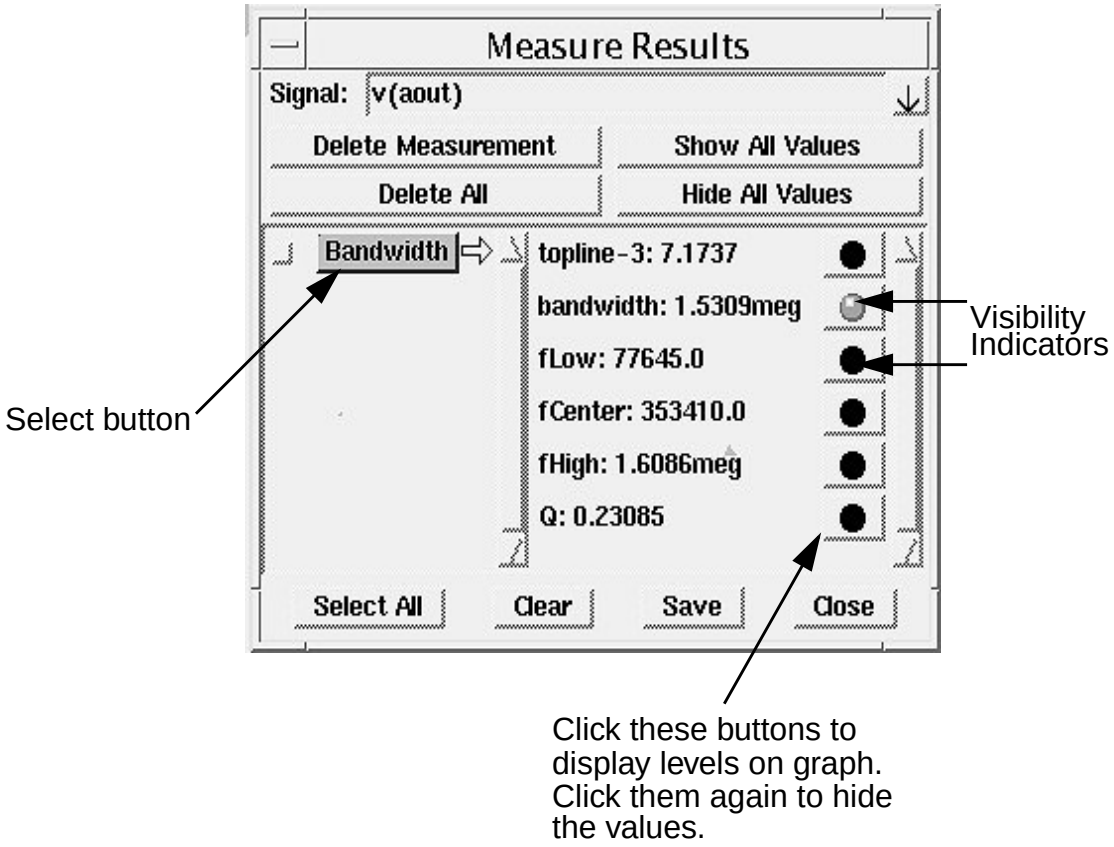
c. You can select the measurement labels and move them if the graph becomes too cluttered. Position the cursor over the text. Then left-click and hold while moving the cursor to a new location.

5. You can get more information about each of the measures you performed or control the amount of information displayed in the Graph window by using the Measure Results dialog box:

   a. In the Graph window, move the mouse cursor to the `v(aout)` signal name.

   b. Use the popup menu and choose the **Signal Menu > Measure Results...** item. A Measure Results dialog box appears.

   c. In the Measure Results dialog box, be sure the Bandwidth item in the left column is selected (see the following figure).



   d. Notice in the Measure Results dialog box, in the right column, the different values that are available from executing the bandwidth measurement.

   e. Click on the various visibility indicators to choose which values are displayed in the Graph window.

   f. When you have finished exploring the Measure Results dialog box, close it.

6. When you have finished trying out the features of CosmosScope, close the application by selecting the **File > Exit** menu item.

This concludes the tutorial.

Chapter 4

CosmosScope Menus Reference

This chapter provides reference information on each of the selections available from the CosmosScope pulldown and popup menus:

- File Pulldown Menu Options
- Edit Pulldown Menu Options
  - Undo
  - Cut, Copy, Paste, Delete
  - Graph Preferences
  - Scope Preferences
- Graph Pulldown Menu Options
- Tools Pulldown Menu Options
- Window Pulldown Menu Options
- Help Pulldown Menu Options
- Popup Menus

Additional menus are associated with the Signal Manager Tool; these are covered in the Signal Manager Chapter of this manual.

File Pulldown Menu Options

The **File** pulldown menu allows you to open existing files, save your work to new files, create new windows, save configuration settings, open the print dialog box, and exit the application.

File>New

This option opens a new graph window formatted as an X-Y axis graph, a Smith Chart graph, or a Polar Chart graph.

File>Open

This option allows you to open an existing plot file, graph, or outline using the Open Files dialog box. Select the Files of Type from the pulldown list, and browse to the location of the file you want to open.

File>Close

This option closes the active graph window or the current design.

File>Save

The Save Graph dialog box allows you to specify a path and file name. In addition, a popup dialog box prompts you to save the graph file in one of the following ways:

- With a copy of the waveforms in the graph.
- With a reference to the plot file from which the waveforms in the graph were plotted.

In the first case, all connection to the plot file is lost. In the second case, the connection to the plot is maintained. Thus, if the graph is reopened it can be automatically updated due to any Replace or Append plot actions specified for the plot file in an analysis.

The Save Outline dialog box allows you to specify a path and file name for an outline. In addition, a Graph Outline popup dialog box allows you to specify several attributes for the saved outline. You can select whether or not to maintain the connection to the plot file in the same way as for a graph outline. You control this by checking (or unchecking) the **Dependencies** checkbutton on the Graph Outline dialog box.

The **File > Save > Plotfile (*.txt)** menu choice allows you to save the selected waveforms into a text file.

File>Export Image

This option opens an Export Image dialog box which allows you to export the contents of an editor window to a file in a variety of graphics formats. CosmosScope can create graphics files in the following formats:

PNG (*.png)	Portable Network Graphics
JPEG (*.jpg, *.jpeg)	
TIFF (*.tiff, *.tif)	Tagged Interchange Format
XPM (*.xpm)	X-Window Pixel map
PCL5 (*.pcl5)	
HPGL2 (*.hpgl2)	HP Graphics Language
Postscript (*.ps, *.eps)	
AutoCad DXF (*.dxf)	
CGM (*.cgm)	Computer Graphics Metafile
BMP (*.bmp)	PC Windows Bitmap
EMF (*.emf) in Windows NT only	Enhanced Metafile

File>Configuration

There are two options, and one setting, available.

- **Save** saves your work surface configuration immediately.
- **Clear** clears any saved configuration you have made in the current session. The next time CosmosScope is invoked your configuration will return to the default settings.
- **Save on Exit** checkbox setting saves your configuration settings upon exiting CosmosScope. To do this, you must start the application from the directory in which your work will be performed. The next time you invoke CosmosScope, these settings will be retained.
- **Save in working directory** saves your work surface configuration into the directory where CosmosScope has been invoked.
- **Save in home directory** saves your work surface configuration into your home directory.

When CosmosScope is invoked, it will try to load the configuration file, `.scopecfg`, from the local directory. If it can't find the file, it will try the home directory.

File>Print

Select this option to open the Print dialog box. To print the current graph, single click on the **OK** button.

File>Printer

This menu item appears in UNIX versions of CosmosScope. It allows you to **Create** a new printer configuration, **Remove** a printer from the printer list, or change the **Properties** of your printers.

File>Exit

This option closes the application.

Edit Pulldown Menu Options

Undo

Undo reverses any database operation you have just completed. This item does not un-delete waveforms, undo measurement manipulations, operate on general windows or UI operations. There is one level of undo. If the **Undo** menu item is stippled or greyed out, it will not operate on your last action.

Cut, Copy, Paste, Delete

These menu options operate on the selected object.

- **Cut** removes a selected object and moves it into a clipboard.
- **Copy** copies a selected object in the active window into a clipboard.
- **Paste** will place whatever is in the clipboard into the active window.
- **Delete** will remove the currently selected items from the window.

Graph Preferences

You may customize the appearance of your graphs, and modify other settings by selecting **Edit > Graph Preferences.** This will bring up the Graph Preferences window. Each tab in this window contains the following buttons:

- **Apply** new preferences to all graph windows immediately. This change is good only for your current CosmosScope session unless you use the **Save** button.
- To save your changes between CosmosScope sessions, click on the **Save** button. You can now exit CosmosScope, return, and retain your new preferences.
- The **Defaults** button sets your preferences to the original CosmosScope default selections.
- The **Reset** button returns the work surface to the settings in place when the current session was opened, or when the last settings were applied with the **Apply** button.
- The **Close** button closes the dialog box and returns you to the work surface.

Graph Tab

The **Graph Tab** allows you to change the colors and fonts used in your graphs.

You may specify color selections for **Foreground**, **Highlight**, **Background 1**, and **Background 2**:

- **Foreground** consists of all displayed text, graph outlines, grids, and markers.
- **Highlight** consists of text and signals, which are displayed as reverse video when selected with the mouse cursor.
- **Background 1** is the background of all of the graph regions.
- **Background 2** is the background of the rest of the graph window.

To change colors:

- Single click on the colored square you want to change. A Color Editor dialog box will be displayed, from which you may select or define new custom colors. The reference material on the Drawing Tool provides additional details on the Color Editor.

To change the style of text:

- Click on the **ABC 123** button. The Font Selection dialog box will be displayed to allow you to change the font settings.

Signal Tab

Signals are the information displayed in the graphs. Each time you add another analog signal, it is displayed in a different color. If your screen colors are mapped to Mono, signals are displayed as a variety of dashed lines. These dashed lines cannot be customized.

The **Add** button allows you to add more Signal Color fields.

The **Delete** button deletes the last Signal Color field.

To change signal colors:

- Single click on the buttons that contain the colored square. The Color Editor dialog box will be displayed, from which you may select or define new custom colors. The reference material on the Drawing Tool provides additional details on the Color Editor.

For digital signals, CosmosScope displays different colors and line styles for different logical states. Users can set the color and line style preferences with this tab. Currently, CosmosScope supports `logic_4`, `std_logic`, and `nanosim_logic` type digital signals.

Display Tab

Legend Location Fields

The legend is the text that appears next to the graph containing the labels of the axes and the names of the displayed signals.

The legend can be configured to appear to the right, bottom, left, or top of the graph, or it can be configured as a floating legend.

Selecting the **Float Button** brings the legend up in its own window. This window can be moved anywhere within the graph window.

- To move the legend window, press and hold on the legend window with the left mouse button.
- Drag the window to its new location and release the mouse button.

Grid Visibility Display Default	The options to set the background grid configuration are: • Display - Specify whether to Hide or Show the background grid. Default is "Hide." • Line Style - Select solid, dashed or dotted grid lines. Default is "Dashed". • Line Width - Allows you to set the width of the grid line. Default is "1." • Line Color - Allows you to set the color of the grid line. Default is "White."
Signal Name Default	When **Leaf** is selected, the signal name in the legend will be displayed as the last text string after the last slash when the signal name is a long path name. When **Full Path** is selected, the entire path name will be displayed.
Signal Line Width Default	Sets the default signal line width.
Default multi-member signal color display	Sets the color mode for multi-member signals. In “Single Colored” mode, all the member curves have the same color. In “Rainbow Colored” mode, the member curves may have multiple colors.
Dynamic Waveform Display	Turn on the Dynamic Waveform Display feature by clicking **On** and enter the interval, in seconds, desired for continuously updating the displayed waveform while a simulation is continuing to run.
Open Dynamic Socket	Setting this to ON will allow updates to the graph display via the socket from a simulator running in debug mode.
Signal Highlight On	• Selecting the **Waveform and legend** button allows you to put the mouse cursor on either the signal displayed in the graph region or the name of the signal in the legend in order to highlight the signal. Mouse response is not as quick as with the **Legend only** option. • Selecting the **Legend only** button allows you to put the mouse cursor on the name of the signal in the legend in order to highlight the signal. Mouse response is quicker than with the **Waveform and legend** option.

Signal Draw Feedback	Sets how often the Redraw Status window is displayed. The default number of data points before the Redraw Status window is displayed is 10,000. The higher the number of data points, the fewer Redraw Status window updates are displayed and the faster the window is redrawn.

XY Tab

Customizing specific to the XY type of graph is allowed through the XY Graph Specific fields.

Digital Trace Height field	Changes the height of a digital signal in the trace graph region.
Analog Trace Height field	Changes the height of an analog signal in the trace graph region.
Bus Display Default buttons	Change the base numeric value (radix) of the value displayed in the trace graph region. This option operates when digital signals are combined into a bus.
Trace Snap buttons	Allow you to turn the trace snap for the digital markers on or off. When **On**, the digital marker will snap to the nearest state change. When **Off**, the digital marker can be placed anywhere in the digital graph.
Analog Paste Buttons	Select where new signals will be placed in the graph window if they are not pasted into an existing graph region. • New signals can be placed in a separate, new, graph region by selecting **New Region**. • New signals can be placed in the trace graph region by selecting **Trace Region**. • New signals can be placed in the graph at the bottom of the first graph region window by selecting **Bottom Region**.

Axes Zoom	When you use the axes zoom feature using the cursor, you can use this preference to either: • Have the zoom display exactly where you positioned the cursor zoom area (**Exact** button). • Have the zoom snap to the nearest tick marks from the cursor-defined position (**Nice Ticks** button). The grid increment definition determines where the zoom will snap as defined in the Axis Attributes dialog box, the Grid Increment field.
Pre-Zoom X axis start and end:	Allows you to set the zoom area of the X axis prior to viewing a waveform. The default start point of the zoom is set to start, which specifies the start time of the simulation. The default end point of the zoom is end, which specifies the last point of the simulation. You can specify relative positions to the start and end point. For example, assume you have a waveform covering a simulation time of 0u to 100u. You can specify `start +20% end -20%` to cause the zoom to display the range from 20u to 80u when the waveform is displayed. You can specify a pre-zoom range using specific constant values. Using the same waveform example of 0u to 100u, if you put `20u` as the start point and `50u` as the end point, the waveform will display as zoomed in to that range. You can also specify relative positions using constant values from the start and end point. Again using a waveform that spans 0u to 100u, if you specify a pre-zoom of `start +20u end -20u`, the range from 20u to 80u appears when the waveform is displayed.
Minimum Region Width Minimum Region Height	Set the minimum size of a region. When multiple signals are plotted in different regions, the size of the region may be smaller than what you want. These settings allows you to control the minimum size.

Default dB Scale	Sets the minimum and maximum values of the dB view for signals. The dB view values could be -Inf or Inf, if the original waveform contains 0 or Inf data points. This setting makes the plot easier to read.

Scope Preferences

Reader Tab

Default File Type	Allows you to select the default file type that will be displayed when opening plotfiles. The options reflect all the file types supported by CosmosScope.
Saber PL Reader	Allows you to set the loading mode of the PL reader. Selecting “Incremental” makes the reader an incremental reader - when a plot file is opened, only the header for the plot will be read. Selecting “Non-Incremental” makes the reader a full reader - when a plot file is opened, the whole data of the plot will be read.
Text Writer/Reader	“Writing Precision” allows you to set the precsion of the data that will be written to the text files. “Name/Unit Separator” allows you to set the separator between the waveform name, unit, and type in the header of a text imput file. The default separator is ‘.
Uncompressed Temporary Directory	Allows you to set the location of the temporary directory used for opening gzipped files.

Signal Manager Tab

This tab allows you to set the preferences for signal list windows. Please refer to “Signal Manager Setup Button Dialog Box” in Chapter 5: Signal Manager” for details.

Measurement Tab

This tab allows you to set the preferences for applying measurements. Please refer to “Setting Measurement Preferences” in Chapter 7: Measurement Tool” for details.

Graph Pulldown Menu Options

Plot

This option allows you to plot selected signals from the signal list into the graph region.

Paste

Places the contents of the clipboard into the graph region.

Graph>Annotate Info Menu Option

The **Graph>Annotate Info** menu option brings up a Text Variables dialog box, which allows you to insert information into the graph window. The information available is the current date, the creation date, and the author of the current graph window.

To insert the current Date, Created date, and/or Author user id, click on the adjacent **Insert** button to place the information on the graph. Once the text has been placed into the graph window, you can move it around in the graph window via drag and drop, or the annotations may be edited with the **Draw** tool.

Graph>Zoom Menu Options

By selecting **Zoom to Fit,** the maximum number of data points will be displayed on the graph, to show the entire range of a signal. All displayed graph regions in the graph window are affected.

Zoom In increases magnification to show increased detail. All displayed graph regions in the graph window are affected.

Zoom Out decreases magnification to show less detail, but more of the graphed information. All displayed graph regions in the graph window are affected.

Graph>Signal Attributes Menu Option

Selecting the **Graph>Signal Attributes** option opens the Signal Attributes dialog box. This dialog box allows you to select any signal displayed in the graph window, change the color of the signal, change the style of the line, add a symbol to the signal, change the symbol width, fill the area under the signal, manipulate the stack region, change the point of view of the signal, and change the signal label.

Signal Field

Allows you to select a signal displayed in the graph window to view or modify its attributes.

To display all available signals in the graph window, single click the left mouse button on the downward pointing arrow at the right of the Signal field.

Color Field

Allows you to change the color of the selected signal.

- To change colors, single click on any one of the color buttons.

Style Field

Allows you to change the appearance of the signal line. Several line styles are displayed. Selecting **None** causes the signal line to disappear. If the signal is also represented by symbols, the symbols will still be displayed.

Line Width

Allows you to change the line width of the signal line.

Symbol Field

For analog signals, the Symbol field allows you to add symbols to display the signal line. Several symbol styles are displayed. Selecting **None** causes the symbols to disappear. If the signal is also represented by a line, the line will still be displayed.

Symbol Size Field

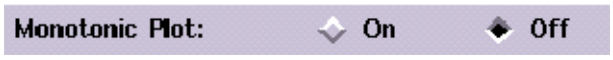
For analog signals, the Symbol Width field allows you to change the size of displayed symbols.

Bar Field

For analog signals, the Bar field allows you to fill in the area under a curve with a pattern. Several bar patterns are displayed. The pattern will be in the color of the signal.

Monotonic Plot Field

For analog signals, the Monotonic Plot field allows you to display the signals in monotonic mode, meaning, only points with interesting x values will be plotted.



Stack Region Field

Allows you to move the signal currently selected in the Signal Attributes dialog box in and out of different graph regions.

- **Digital** will move the selected analog or digital signal into a single digital/trace graph region.
- **New** will move the selected analog or digital signal into a new analog graph region at the top of all other analog graph regions in the graph window.
- *n* (where *n* is the identifying number of a graph region) will move the selected analog or digital signal into the analog graph region specified by the number *n*.

Trace Height Field

For digital signals, the Trace Height field allows you to change the height of selected signals.

Bus Field

For digital signals, the Bus field changes the digital information display.

View Field

For analog signals, the View fields allows you to change the display of the selected signal.

- The left hand field changes the Y-axis (vertical axis) plot of the selected signal.
- The right hand field changes the X-axis (horizontal axis) plot of selected signals based on different X-axis and Y-axis parameters.

Details on the Vertical Axis Options and Horizontal Axis Options are provided following this table, below.

Label Field

Allows you to change the name of the signal.

- To change the name type the name of the signal in the Label field.
- Click the **Apply** button to display the new name in the graph window.

Close Button	Closes the dialog box and returns you to the work surface.

Signal Attributes - View Axis Options

Various options for the Vertical and Horizontal Axis are available by selecting **Graph>Signal Attributes**; these options are implemented by setting the View: fields in the Signal Attributes Dialog Box. Details on these options are provided below.

Vertical Axis Options

- **Real(x)** uses the real X values of a signal as the Y-axis of the graph region. For example, if the X-axis is represented in Hertz, the Y-axis will also be represented in Hertz.
- **dB(x)** uses the decibels of the x value of a signal as the Y-axis .
- **Real(y)** plots the real value of the Y-axis.
- **dB(y)** plots the Y-axis in decibels.
- **Mag(y)** plots the magnitude of the Y-axis value.
- **Imag(y)** plots the Y-axis in imaginary terms.
- **Phase(y) deg** plots the Y-axis values .
- **Phase(y) rad** plots the Y-axis in terms of phase in radian units.
- **CPhase(y) deg** plots the Y-axis in terms of continuous phase, in degree units.
- **CPhase(y) rad** plots the Y-axis in terms of continuous phase, in radian units.
- **dBm(y)** plots the Y-axis values in decibels, times 1,000.
- **dBu(y)** plots the Y-axis values in decibels, times 1,000,000.

Horizontal Axis Options

- **Real(x)** plots the real value of the X-axis.
- **dB(x)** plots the X-axis values in decibels.
- **Real(y)** uses the real Y values of a signal as the X-axis of the graph region. For example, if the Y-axis is represented in volts, the X-axis will also be represented in volts.

- **dB(y)** plots the X-axis in Y-axis terms and decibels. For example, if the Y-axis is represented in volts, the X-axis will be represented in volts and decibels.
- **Mag(y)** plots the magnitude of the Y-axis value.
- **Imag(y)** plots the X-axis in imaginary Y-axis terms. For example, if the Y-axis is represented in volts, the X-axis will be represented in imaginary volts.
- **Phase(y) deg** plots the X-axis in terms of phase, in degree units.
- **Phase(y) rad** plots the X-axis in terms of phase, in radian units.
- **CPhase(y) deg** plots the X-axis in terms of continuous phase, in degree units.
- **CPhase(y) rad** plots the X-axis in terms of continuous phase, in radian units.
- **Real YvsX** plots the signal with the Y-axis vertically and the X-axis horizontally and in real terms.
- **Real XvsY** plots the signal with the X-axis vertically and the Y-axis horizontally and in real terms.
- **dBm(x)** plots the X-axis values in decibels, times 1,000.
- **dBu(x)** plots the X-axis values in decibels, times 1,000,000.

Graph>Axis Attributes Menu Option

This option opens the Axis Attributes dialog box. This dialog box allows you to select an axis, zoom, pan, change the scale to linear or logarithmic, change grid increments, toggle the axis grid, and edit the axis label.

Axis Field

Allows you to select and update any axis displayed in the graph window. To display and select an available axes from the graph window, click the left mouse button on the down arrow to the right of the Axis field.

Range Fields

These fields allow you to zoom and pan along the axis associated with the Axis field. The field of view is displayed in the Zoom From/To field. The entire possible viewing range is displayed beneath the **Slider** bar.

Zoom In Button

Allows you to increase magnification to show increased detail.

Zoom to Fit Button

Allows you to display the maximum number of data points to show the entire range of a signal.

Zoom Out Button

Allows you to decrease magnification to show less detail, but more of the graphed information.

Zoom From/To Field

Displays the current field of view for the selected axis and allows you to enter values for the range you want to display.

- To change the field of view, type in the values in the From/To fields and Single click on the **Apply** button.

Slider Bar

Allows you to pan along the selected axis.

- Press and hold the left mouse button and move the slider bar right or left. The range available to be displayed is shown below the slider bar. The range displayed is dynamically shown in the From/To field.
- Release the mouse button when the coordinates you want are displayed in the From/To field.

Unit Scale Factor

Allows you to set the unit prefix for axis labels. For example, selecting 'm' will cause all labels to be in "mili-" scale. The default is "auto," and it means not specified.

Scale Field

Toggles the selected axis scale. The **Linear** button displays the selected axis scale in linear increments. The **Log** button displays the selected axis scale in logarithmic increments.

Increment Field (Grid, Exponent)

Allows you to change the resolution of the axis scale.

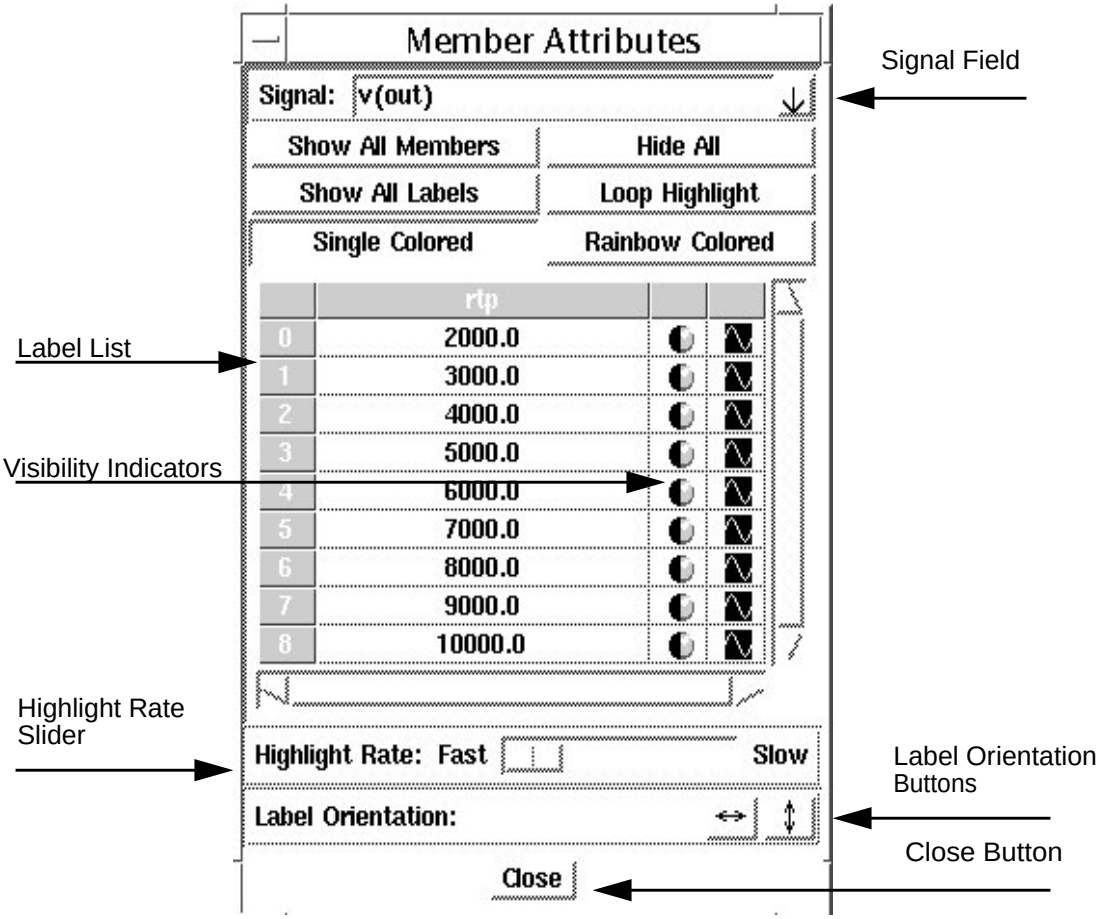
If you selected **Linear** for Scale, you will set the Grid Increment. If you selected **Log** for Scale, you will set Exponent Increment and Grids/Increment.

- Grid Increment -- Sets the unit difference between labeled grids for a linear scale. For example, inserting 20u for a time scale will set each grid marking 20usec apart. This setting is also used by the Nice Ticks option in the Axes Zoom field of the Graph Preference dialog box, the XY tab.
- Exponent Increment -- Sets the unit difference between labeled grids for a log scale. For example, inserting 1 for a frequency scale will set each grid marking exponentially apart—10, 100, 1000, etc.
- Grids/Increment -- Sets the number of unlabeled subdivisions between grids for a log scale.

Number of Tick Subdivisions Field	Allows you to change the number of gradations (ticks) between grid increments. • To change the number of gradations, type in the number of ticks you want displayed and single click on the **Apply** button.
Grid Visibility Field	Toggles the grid display. The **Show** button turns the axis grid in the active graph window on. The **Hide** button turns the axis grid off.
Axis Label Field	Allows you to change the label of the axis. The actual units of the scale are not changed. • To change the axis label type the name of the signal in the Axis Label field. • Click the **Apply** button to display the new name in the graph window.
Close Button	Exits the dialog box and returns you to the work surface.

Graph>Members Menu Option

This Graph menu option opens the Member Attributes dialog box, if a graph window containing multiple members is open. This allows you to control the display of a signal consisting of multiple members, such as those generated by Vary or Monte Carlo data analysis. You can show all members or an individual member, hide all members or an individual member, show all member labels or an individual member label, and loop through all members of the signal, highlighting each member individually.



Label List

Displays the label associated with each member of a signal. Each label is associated with a visibility indicator to the right in the dialog box.

Signal Field

Allows you to select any signal displayed in the graph window.

- To display all available signals in the graph window single click with the left mouse button on the downward pointing arrow at the right of the Signal field.

Visibility Indicator

Each **Visibility Indicator** shows the display status of a member. Each indicator is associated with a member label in the **Label List** to the left.

- To change the state of the visibility indicator, single click with the left mouse button.
- An all black visibility indicator shows that neither the label nor the member of the signal is displayed.
- A half black and half green visibility indicator indicates that the member of the signal, but not the label, is displayed.
- An all green visibility indicator indicates that both the label and the member of the signal are displayed.

Color indicator

Each **Color indicator** shows the color of a member. Clicking on it with the left mouse ubtton allows you to change the member's color.

Show All Members Button

Displays all members of the signal shown in the Signal field. No labels are displayed.

Show All Labels Button

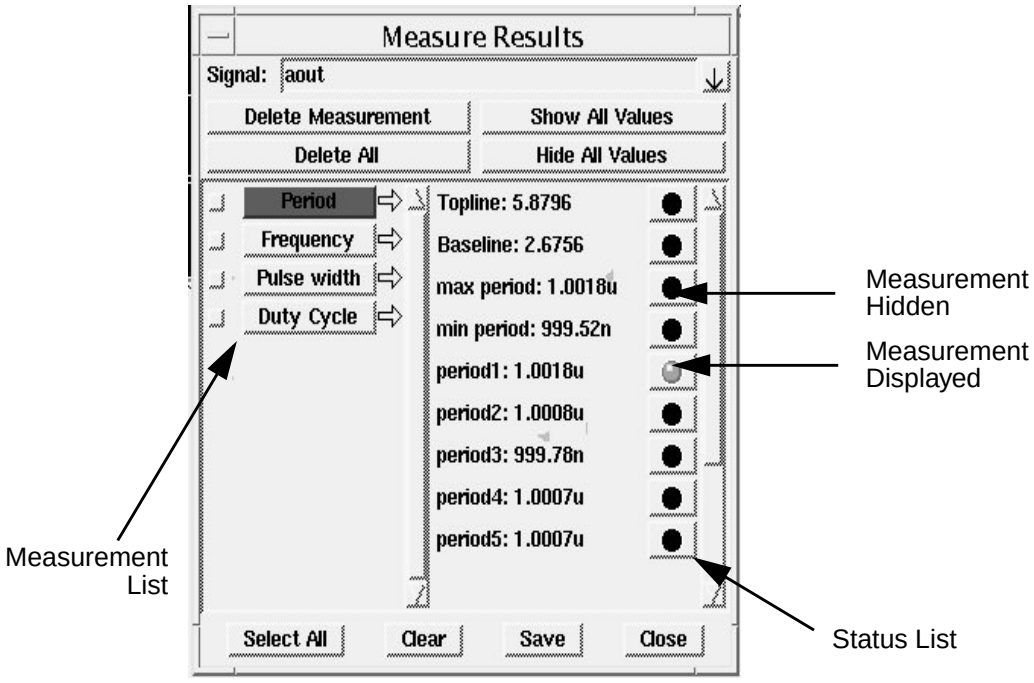
Displays all members, with labels, in the graph window. The location of these labels can be rearranged for ease of viewing.

- To move a label, press and hold the left mouse button on a label you wish to move and move the label.
- Release the mouse button to place the label.

Hide All Button	Removes all members in the graph window from view. Individual members can be displayed by using the visibility indicators or all members can be displayed by using the **Show All Members** button.
Loop Highlight Button	Sequentially highlights each member of a signal. This button functions even if all members have been hidden. • To start or stop the loop, single click on the **Loop Highlight** button
Single Colored Button	Sets the color mode of the multi-member wavefrom to “single colored.” All the members will have the same color.
Rainbow Colored button	Sets the color mode of the multi-member wavefrom to “rainbow colored.” The members may have different colors.
Highlight Rate Slider	Regulates the rate of the **Loop Highlight**. When the slider is set all the way to **Slow** there is a 5 second delay between the highlight of members. The resolution for control of the delay is in 50 millisecond increments.
Label Orientation Buttons	Stack member labels from top to bottom or from left to right.
Close Button	Closes the Member Attributes dialog box.

Graph>Measure Results Menu Option

The Measure Results dialog box allows you to select a signal, delete measurements, hide measurement results, or show all measurement results.



The **Measurement** list displays all measurements that are active for a signal. Measurement values for the selected measurement are displayed in the **Status** list.

Status List

Displays a list of measurement values associated with the selected measurement in the **Measurement** list. Each value has a visibility indicator associated with it.

Visibility Indicator

A **Visibility Indicator** shows the display status of a measurement value.

- To change the state of the visibility indicator, single click the left mouse button.
- An all black visibility indicator indicates that the measurement value is not displayed.
- An all green visibility indicator shows that the measurement value is displayed.

Signal Field	Allows you to select any signal displayed in the graph window. To display all available signals in the graph window, single click with the left mouse button on the downward pointing arrow at the right of the Signal field.
Select checkbox	The check boxes before the measurements are used to select the measurements that you want to export to a text report file.
Delete Measurement Button	Permanently deletes the selected measurement.
Delete All Button	Permanently deletes all measurements for the selected signal.
Show All Values Button	Displays all values for the selected measurement.
Hide All Values Button	Hides the display of all values for the selected measurement. Passing the mouse cursor over a visibility indicator in the Status list will cause a highlighted measurement value to appear on the signal.
Select All	Selects all the measurements for exporting to a text report file when the **Save** button pressed.
Clear	Deselects all selected check boxes.
Save	Selecting this button will pop up a dialog window with information for the selected measurements. Information will be saved to a text file. The **Save** button is found at the bottom of the window.
Close Button	Closes the Measure Results dialog box.

Graph>Waveform Compare Menu Option

Selecting this menu choice opens the Waveform Compare dialog box. This feature reports differences between waveforms of two selected signals. Select the graph you wish to compare from the Graph List pull-down list; highlight the **Signal1** button, and double click on the desired signal in the Signal List. Select the **Signal2** signal (which may be from a different Graph) in a similar manner. Then, specify the desired Compare Options. Click on the Apply button to generate the result. This feature applies to digital signals only.

Graph>Signal Search Menu Option

To locate the Graph and Signal numbers in which a specified signal appears, enter the name of the signal in the Signal Search dialog box, and click on the **Search** button

Graph>Selected Axes Menu Option

Operates on one or more axes that you have selected. **Selected Axis** items that are available to alter or show/hide are **Range**, **Scale**, **Grids**, **Sliders**, and **Lock**.

To select an axis, place the mouse cursor over the axis grid, the axis name next to the axis grid or the axis name in the legend, and single click with the left mouse button.

Range

The following options are available by selecting **Graph>Selected Axes>Range:**

- **Zoom to Fit** displays the maximum number of data points to show the entire range of a signal.
- **Zoom In** increases magnification to show increased detail.
- **Zoom Out** decreases magnification to show less detail, but more of the graphed information.
- **Pan Right** allows you to view parts of the data that are not visible to the right of the currently displayed graph. This command operates only on the X-axis.
- **Pan Left** allows you to view parts of the data that are not visible to the left of the currently displayed graph. This command operates only on the X-axis.

- **Pan Up** allows you to view parts of the data that are not visible above the currently displayed graph. This command operates only on the Y-axis.
- **Pan Down** allows you to view parts of the data that are not visible below the currently displayed graph. This command operates only on the Y-axis.

Scale

The following options are available by selecting **Graph>Selected Axes>Scale:**

- **Linear** sets the selected axis to a Linear scale.
- **Log s**ets the selected axis to a Logarithmic scale.

Grids

- **Show** turns the axis grid in the graph regions on.
- **Hide** turns the axis grid in the graph regions off.

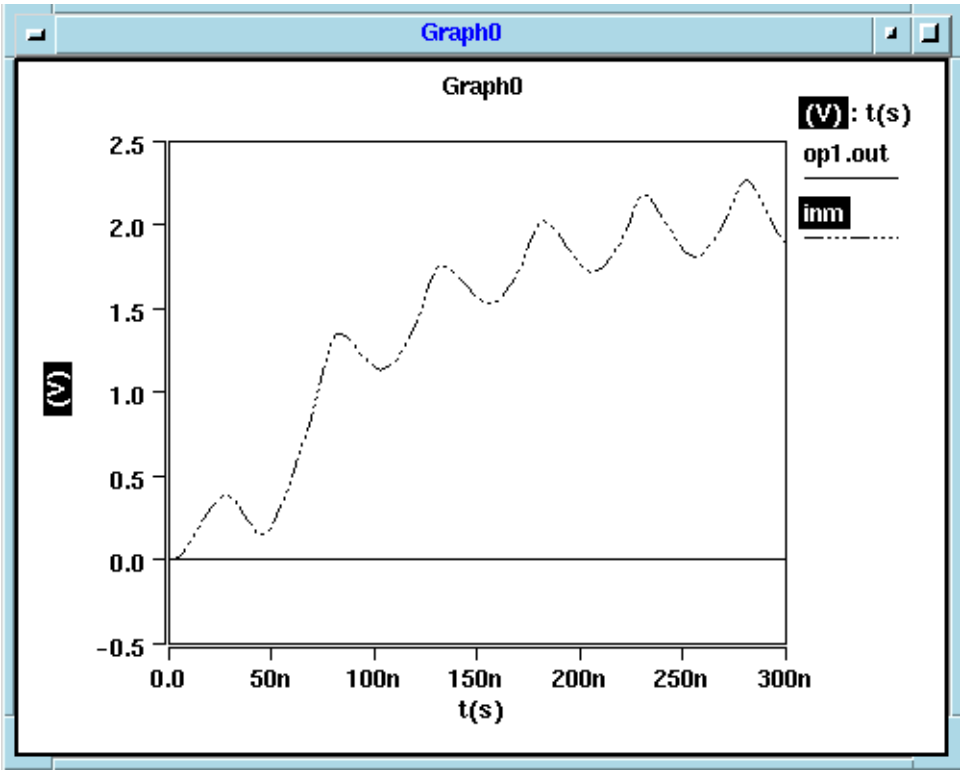
Sliders

You have the option of using a slider bar to pan along the X-axis of a graph.

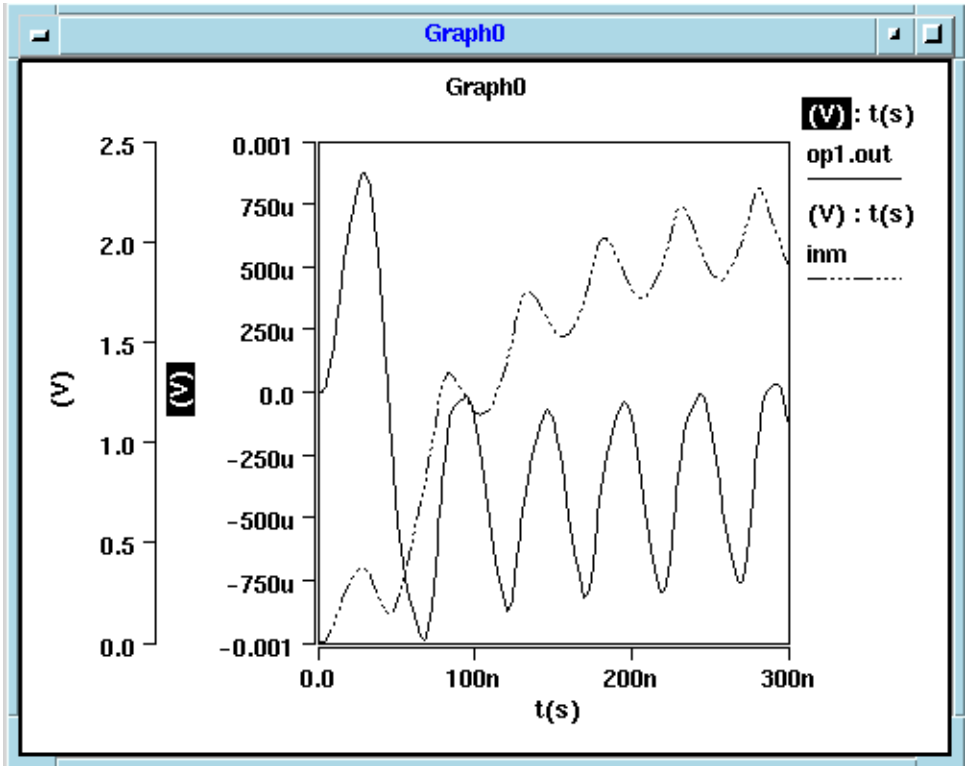
- **Show** turns the slider bar on.
- **Hide** turns the slider bar off.

Lock Menu Item

Accept new signals allows signals to be added to the axis if the signals share the same units. For example, a signal with an amplitude in microvolts will share the same scaling as a signal with an amplitude in volts in the same graph region.



Lock out new signals does not allow signals to be added to the same axis even if they have the same units. For example, a signal with an amplitude in microvolts will have a different scale from a signal with an amplitude in volts in the same graph region.



Graph>Selected Signals Menu Option

The Selected Signals feature operates on one or more graphed signals that you select, allowing you to adjust the display characteristics.

Selected Signals items that are available are **Stack Region**, **Color**, **Style**, **Line Width**, **Symbol**, **Symbol Width**, **View**, **Signal Grid Trace Height**, **Digital Display**, **Create Bus**, **Convert to Digital**, and **Delete Signals**.

To select a signal, place the mouse cursor over the signal or over the signal name in the legend, and single click the left mouse button.

Selected Signals>Stack Region Menu Item

Allows you to move selected signals in and out of different graph regions.

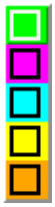
Digital/Trace moves selected analog or digital signals into the digital/trace graph region.

New Analog moves selected analog or digital signals into a new analog graph region.

Analog *n* (where *n* is the identifying number of a graph) moves selected analog or digital signals into the analog graph region specified by the number *n*.

Selected Signals>Color Menu Item

Allows you to choose one of the available signal colors.

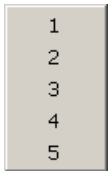


Selected Signals>Style Menu Item

Displays several line styles. Selecting **None** causes the signal line to disappear. If the signal is also represented by symbols, the symbols will still be displayed.

Selected Signals>Line Width

Allows you to change the line width of the signals.

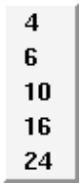


Selected Signals>Symbol Menu Item

Displays several symbol styles. Selecting **None** causes the symbols to disappear. If the signal is also represented by a line, the line will still be displayed.

Selected Signals>Symbol Size Menu Item

Allows you to change the size of displayed symbols.



Selected Signals>View Menu Item

Allows you to change the display of selected signals

Horizontal Value changes the X-axis (horizontal axis) plot of selected signals based on different X-axis and Y-axis parameters.

Vertical Value changes the Y-axis (vertical axis) plot of selected graph regions.

AC signals will have the option to change the display to a **Nyquist** plot, **Bode dB** plot (the vertical scale is represented in decibels), **Bode Ph** (phase) plot (the vertical scale is represented in degrees), or a **Nichols** plot.

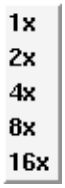
Monotonic Plot allows you to switch to plot of selected signals between monotonic mode and normal mode.

Selected Signals>Signal Grid Menu Item

This features allows you to add a grid for selected (digital) signals. Select one or more signals from the graph window, then select the Signal Grid Menu item.

Selected Signals>Trace Height Menu Item

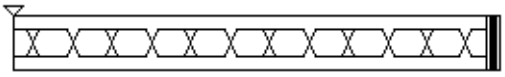
Valid for digital or analog signals displayed in the Digital/Trace graph region. The **Trace Height** menu allows you to change the height of selected signals.



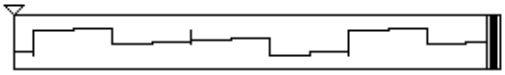
Selected Signals>Digital Display Menu Item

Allows you to display logical information in numerical forms such as binary, octal, decimal, hex, double, and stepped. For decimal, double, and stepped display, the selections **unsigned**, **One's Complement**, and **Two's Complement** are available.

Digital signals combined into a bus can be displayed as a doubled waveform,



or as an analog stepped waveform.



Selected Signals>Create Bus Menu Item

Takes multiple digital signals in the trace graph region and combines them into a single digital bus.

To use the **Create Bus** item you first need to have two or more digital signals that you want to combine into a bus.

- Hold the **Shift** button on the keyboard and select each signal with the left mouse button.
- The first signal selected is the least significant (LSB) and each additional signal selected is more significant until the last signal selected, which is assumed to be the most significant (MSB).

- Click on the **Create Bus** icon or menu item and the signal is plotted in CosmosScope and given the designation Group*number*.

Some characteristics of this signal can be edited with the Signal Attributes dialog box. The default display type is Hexadecimal, but Octal or Binary can be selected and the label can be changed from Group*number* to another label designation.

- To open the Signal Attributes dialog box double click on the bus label.

Selected Signals > Convert To Digital

Allows you to input the low and high thresholds, then creates and plots digital signals converted from the selected analog signals.

Selected Signals > Delete Signals Menu Item

Deletes the signal or signals that are selected in the graph window. If the deleted signal is the only signal in the graph region, the whole graph region is deleted.

Graph>Selected Graphics Menu Option

The **Graph>Selected Graphics** menu choice provides two options:

When **Selected Graphics > Manage** is selected, the selected graphic objects will expand or shrink as the graph is zoomed in or zoomed out. Un-managed graphic objects will remain the same size.

The **Delete** option will remove any Draw graphics selected in the graph window. Select the graphic by highlighting it with the cursor and clicking the left mouse button. Multiple graphics can be selected by holding down the **Shift** key. Then, click on the **Selected Graphics > Delete** item.

Graph>Font Menu Option

This menu option controls the font selection, style, and size displayed in the current graph.

Graph>Color Map Menu Option

This option allows you to change the graph window color scheme to **Map 1**, **Map 2**, and **Mono**. **Map 1** and **Map 2** are color maps for color displays and color printers. **Mono** is a black and white display for monochrome displays and black ink printers.

All of the color maps can be altered in the **Edit > Graph Preferences** dialog box.

Graph>Legend Menu Option

The **Legend** option allows you to change the position of legends in the graph regions. The legend is the text that appears next to the graph containing the labels of the axes and the names of the displayed signals.

The legend can be configured to appear to the Right, Bottom, Left, or Top of the graph, or it can be configured to Float.

Selecting the None item removes the legend from the graph window. The legend can be redisplayed by using any of the other Legend items.

Graph>Match Aspect Ration Menu Option

This option allows you to change the dimensions of the graph window. This can be useful for matching the aspect ratio of a sheet of paper for printing hardcopy.

To change the graph window dimensions to match the aspect ratio, type the values for height and/or width in the appropriate fields and click the OK button.

Graph>Rename Window Title Menu Option

This option allows you to rename the title on the graph window.

Graph>Clear Graph

Selecting this option will clear all signals, axes, and legends from the graph window, without saving the information.

Tools Pulldown Menu Options

The Tools menu provides alternative access to the various tools that are available as Tool Bar Icons at the bottom of the screen. Refer to the documentation set for these tools: Command Line Tool, Draw Tool, Signal Manager, Measurement, Calculator, Macro Recorder, RF Tool, and MATLAB.

Window Pulldown Menu Options

The Windows pulldown menu allows management of multiple windows on the work surface. All of the windows on the work surface are listed at the bottom of the Windows pulldown menu.

To open an iconized window or to bring a window to the front of a stack of windows, single click the left mouse button on the window name in the list.

Other functions are:

Cascade	Places windows so that they are on top of each other diagonally from the upper left hand corner of the work surface to the lower right hand corner of the work surface.
Tile	Places windows so they have no overlap.
Arrange Icons	Neatly arranges multiple icons across the bottom of the work surface.

Help Pulldown Menu Options

The Help pulldown menu provides access to the **Online Documentation** help files. The **About** options display information about the current version of the software, and provides the option to display the Copyright/Legal information for the product.

CosmosScope Popup Menus

To open a popup menu:

- Highlight a signal, axis, legend, marker, graphic, or measurement.
- Then press and hold the right mouse button.

To activate an item in the popup menu:

- Place the mouse cursor over the item, and release the mouse button.

Popup menus available in the graph window:

Trace Popup Menu

Graph Popup Menu

Axis Popup Menu

Signal Popup Menu

Measure Popup Menu

AimDraw Popup Menu

Trace Popup Menu

To open the **Trace** popup menu, place the mouse cursor over the Marker field or a Trace marker, then press and hold the right mouse button.

The **Trace** popup menu allows you to display the logic level of a specific Trace marker, to center the associated axis on a specific Trace marker, to display the absolute and/or relative x values on the markers, and to turn the marker snap-to-trace behavior on or off.

Display Brings up a list of active Trace markers. Selecting a Trace marker from the list displays the logic state of the digital signal, and the value of an analog signal in the Trace graph region, at the location of the Trace marker. The logic state is displayed in the Marker field.

Center Brings up a list of active Trace markers. Selecting a Trace marker from the list pans the associated axis so that the marker is centered in the Trace graph region.

Delete Brings up a list of active Trace markers. Selecting a Trace marker from the list deletes that marker from the Trace graph region.

Value Type Brings up a list of **absolute**, **relative**, or **both**. Selecting an item from the list sets the value types of the x-values shown on the top of the markers. The relative values are shown in parentheses.

Ref Marker Brings up a list of active Trace markers. Selecting a Trace marker from the list sets the marker as the reference marker. The relative x value on the reference marker is `0.0`.

Snap to Trace Allows you to make Trace markers snap to discrete events on a digital signal.

Graph Popup Menu

This menu can be displayed by selecting the CosmosScope **Graph** pull-down menu option (described previously), or as a popup:

To open the Graph popup menu:

- Place the mouse cursor in a blank part of the graph window, so that no object is highlighted.
- Then press and hold the right mouse button.

To activate an item in the Graph popup menu:

- Place the mouse cursor over the item you wish to open and release the right mouse button.

Axis Popup Menu

To open the Axis popup menu:

- Place the mouse cursor over an axis or axis name.
- Then press and hold the right mouse button.

The **Axis** popup menu allows you to manipulate the horizontal and vertical scales of the axes in the graphs.

Attributes dialog box	Allows you to select an axis, zoom, pan, change the unit scale factor, change the scale to linear or logarithmic, change grid increments, toggle the axis grid, and edit the axis label.
Range menu	Allows you to perform zooming and panning.
Scale menu	Allows you to display the axis scale in **Linear** or **Log** increments.
Display Grid button	Turns the axis grid for the selected axis on or off.
Display range slider	Button displays or hides the X-axis slider bar. You have the option of using this slider bar to pan along the X-axis of a graph.
Lock out new signals button	Does not allow signals to be added to the same axis even if they have the same units.

Signal Popup Menu

To open the Signal popup menu:

- Place the mouse cursor over the signal or signal name.
- Then press and hold the right mouse button.

The **Signal** popup menu allows you to manipulate the display of signals in the graphs.

Attributes
Opens the Signal Attributes dialog box which allows you to select any signal displayed in the graph window, to change the color of the signal, to change the style of the line, to change the width of the line, to add a symbol to the signal, to change the symbol width, to fill the area under the signal, to manipulate the stack region, to change the point of view of the signal, and to change the

signal label. This dialog box was described under the **Graph>Signal Attribute**s topic, above.

Members
Opens the Member Attributes dialog box which allows you to control the display of a signal consisting of multiple members such as those generated by Vary or Monte Carlo data analysis. You can show all members or an individual member, hide all members or an individual member, show all member labels or an individual member label, set the colors for individual members, and sequentially highlight members over the entire signal range. This dialog box was described under the **Graph>Members** topic, above.

Measure Results
Opens the Measure Results dialog box allows you to select a signal, delete measurements, hide measurement displays or show all measurement results, save measurement results to a text file.

Annotate Info
Brings up a Text Variables dialog box. The Text Variables dialog box allows you to view information on the displayed signals and to insert that information into the graph window.

Stack Region, **Color**, **Style**, **Line Width**, **Symbol**, **Symbol Size**, **View**, **Trace Height**, **Digital Display, Convert to Digital**
These menu options operate in the same manner as the menu options previously described under the CosmosScope **Graph>Selected Signals** menu.

Burst Bus
Expands a bus into discrete digital signals.

Contract Bus
Contracts the extracted bus. The inverse of “Burst Bus.”

Signal Grid
Toggles the signal grid on selected signals. This applies to digital signals.

To Time Domain
Converts a frequency domain signal to a steady state time domain signal.

Front
Places the selected signal at the front of a stack of overlapping signals for display.

Delete Signal
Deletes a highlighted signal. If the deleted signal is the only signal in the graph region, the whole graph region is deleted as well.

Measure Popup Menu

To open the Measure popup menu:

- Place the mouse cursor over the Measurement marker.
- Then press and hold the right mouse button.

The **Measure** popup menu allows you to open the Measure Results dialog box, hide measurement values, lower a measurement label to the bottom of a stack of labels, and add markers to the signal.

Measure Results dialog box
Allows you to select a signal, delete measurements, hide measurement displays, or show all measurement results.

Hide Measurement
Turns off the measurement value on the measurement that the **Measure** popup menu is acting on. The value can be redisplayed by using the Measure Results dialog box.

Delete Measurement
Deletes the selected menu.

Back
Sends the measurement label to the back of a stack of labels for display.

Front
Sends the measurement label to the front of a stack of labels for display.

Add Marker
Can add three different kinds of measurement markers to a signal.

- A **Point Marker** duplicates an existing anchor point and label.
- A **Horizontal Level** marker creates a horizontal line that intersects the vertical (Y-axis) value of an anchor point on a signal.
- A **Vertical Level** marker creates a vertical line that intersects the horizontal (X-axis) value of an anchor point on a signal.

AimDraw Popup Menu

To open the AimDraw popup menu:

- Place the mouse cursor over a Draw graphic.
- Then press and hold the right mouse button.

The **AimDraw** popup menu allows you to edit graphics in a window. You can change the attributes of a selected graphic object, rotate it, flip it, put it at the back of a stack, put it at the front of a stack of, or delete it.

Attributes
Opens an AimDraw Attributes dialog box. This dialog box allows you to change the characteristics of the selected graphic object such as color, line width, fill, font, etc. Each graphic object has a slightly different Attributes dialog box.

Rotate
Allow you to rotate a graphic object **90**, **180**, or **270** degrees.

Flip
Allow you to mirror image a graphic object horizontally, by choosing the **Up-Down** item, or vertically, by choosing the **Left-Right** item.

Back
Lowers a graphic to the bottom of a stack of objects.

Front
Raises a graphic to the top of a stack of objects.

Delete
Permanently deletes the graphic that the **AimDraw** popup menu is acting on.

Chapter 5

Signal Manager

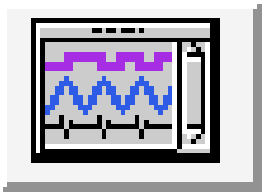
The Signal Manager is a tool for managing the signals generated by a design analysis. The Signal Manager allows you to open plotfiles, filter out unwanted signals, place signals into a CosmosScope graph window, and place signals into the Calculator.

The Signal Manager is described in the following topics:

- Accessing the Signal Manager
- Opening a Plotfile
- HSPICE Sweep Filtering
- Searching Multiple Plotfiles for Signals
- Signal Manager Dialog Box
- Signal Manager Menus
- Signal Manager Buttons
- Signal Manager Plotfile Window

Accessing the Signal Manager

The Signal Manager icon is located in the Tool bar at the bottom of the work surface.



To open or close the Signal Manager

- Single click on the icon with the left mouse button.

Opening a Plotfile

1. To open a Plotfile, single click on the Signal Manager icon. The Signal Manager Dialog Box will be displayed.
2. Single click on the **Open Plotfiles...** button to display the Open Plotfile dialog box.
3. Select the directory where your plotfiles reside from the Directories scrollable list. Select the desired Files of Type entry, and all such matching plotfile names will appear in the File Name scrollable list.
4. Double click on the plotfile name to display the plotfile window.
5. Single click on the signals you wish to plot. Deselect a highlighted signal by single clicking it again.
6. Plot the signals to the current graph window by single clicking the **Plot** button.

You can also open plotfiles and bring up the Signal Manager by using the **File > Open > Plotfiles...** item in the menu bar or by single clicking on the Open icon in the CosmosScope icon bar.

Copying a Signal into the Calculator

- To copy a signal into the calculator, highlight the signal in the plotfile window; place the mouse cursor in the calculator entry field, and click the middle mouse button.

Copying a Signal into Saber Simulator Guide

- When using either CosmosScope or Saber, copy a signal into a Guide form by highlighting the signal in the plotfile window, placing the mouse cursor in the form field, and clicking the middle mouse button.

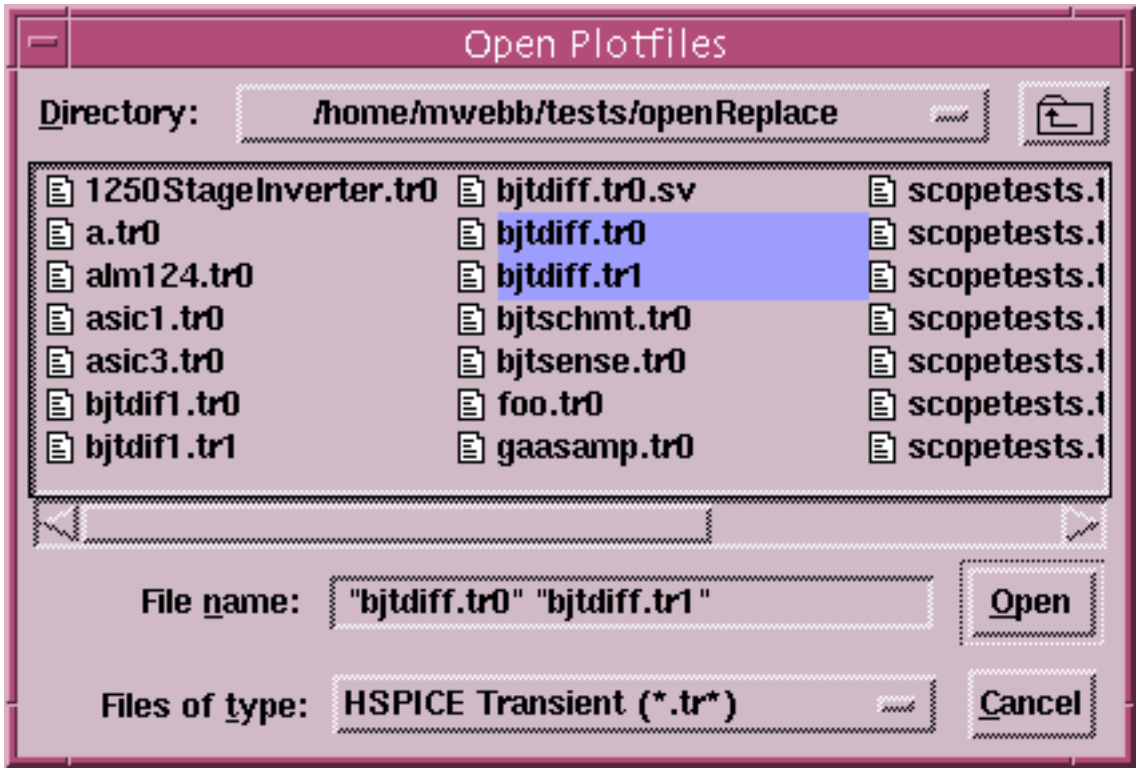
HSPICE Sweep Filtering

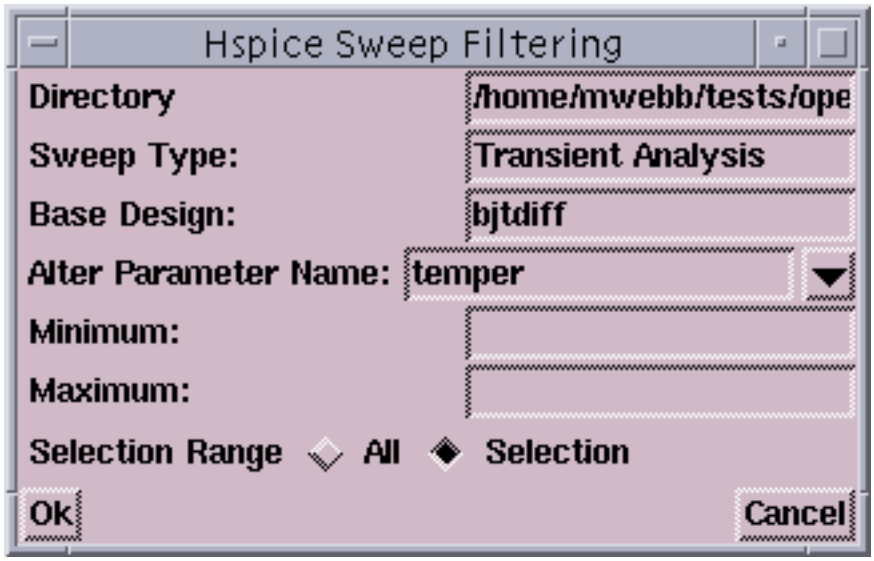
It is possible to have a single HSPICE run produce multiple output files. For example, you can set CosmosScope to produce multiple transient analysis files for each temperature setting with temperature as the "Parameter."

To use this feature in the signal manager, click "**Open Plotfiles**." In the "**Open Plotfiles**" dialog, select two or more analyses output files with the same root name and the same type of analysis (*e.g.*, bjtdiff.ac0, bjtdiff .ac1). If the measurement analysis files are available (in this case, bjtdiff.ma0, bjtdiff.ma1), the tool will permit you to specify the range of parameters you are interested in. (Select 2 or more runs from the same HSPICE execution .tr0 .t1.)

The Hspice Sweep Filtering dialog will appear. Alter Parameter Name, Minimum, and Maximum will become enabled when **Selection Range** is set to **Selection**. (To select **All** output files, you must select two or more analyses first.)

The pull down menu will let you select sweep parameter or measurement. Minimum:/Maximum: will let you specify the range to open. Click **OK**, and the plotfiles whose parameters fall within that range will be opened and displayed in the signal list dialog.





Searching Multiple Plotfiles for Signals

Saber and CosmosScope provide a way to search several waveform plotfiles for signals that may be of interest. For example, you may want to display signals for comparison from several different plotfiles. To do this, follow these steps:

1. Click the Signal Manager icon button on the lower part of the display.
2. In the Signal Manager window that appears, click **Open Plotfiles**.
3. In the Open Plotfiles window, navigate to the directory containing a waveform plotfile(s) of interest and select a plotfile.
4. From the swept results of a single design, type a pattern name on the Filter edit line.
5. Click the **Filter** down arrow to display the Filter Attributes menu.
6. In the Filter Attributes menu, select the desired filter attribute(s).
7. Signal names corresponding to your filter attributes will be highlighted in the File Name window.
8. You may plot any or all of these signals.

NOTE

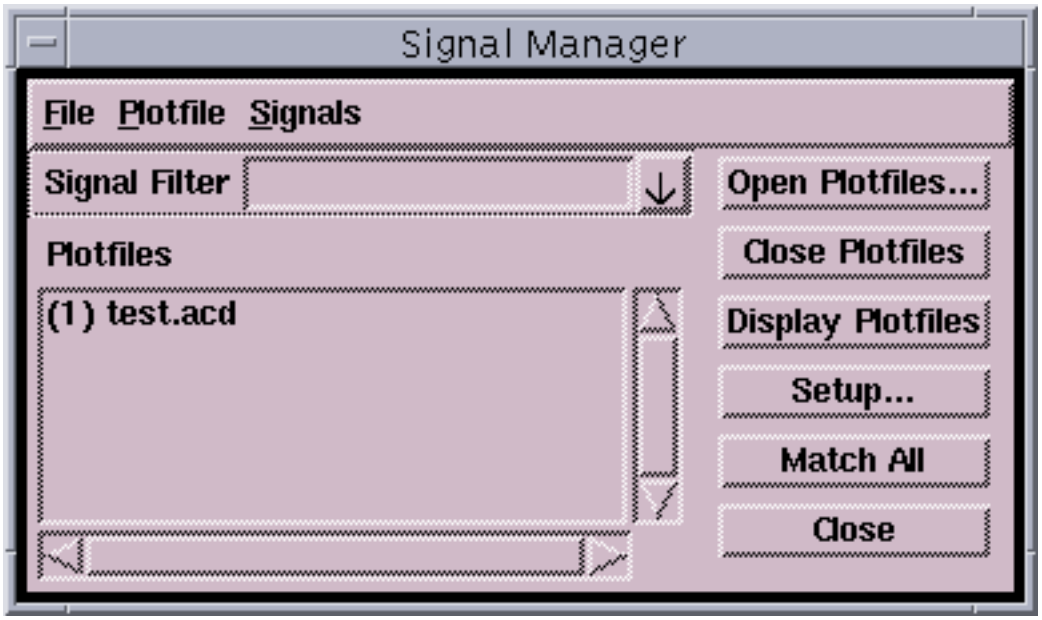
When you enter search criteria on the Filter edit line in the File Name window, regular expression syntax may be used to specify the search parameters.

Signal Manager Dialog Box

The Signal Manager dialog box allows you to open multiple plotfiles, and manage how they are displayed.

Open plotfiles are displayed in the Plotfiles scrollable list box. Plotfiles are numbered sequentially in the order that they were opened.

If a plotfile contains multi-member signals, this count will be reflected in brackets to the right of the signal name.



Signal Manager Menus

The following menus are available from the Signal Manager dialog box.

File Menu	Manages opens, loads, and closes plotfiles.
Plotfile Menu	Manages plotfiles displayed in the Plotfiles scrollable list.
Signals Menu	Plots or deselects selected signals in a plotfile.

Signal Manager File Menu Items

The following items appear in the **File** menu list in the Signal Manager dialog box:

Open Plotfiles	Opens the Open Plotfiles dialog box allowing you to open new plotfiles. (You can also open plotfiles by single clicking on the **Open** icon in the icon bar.)
Reload All Plotfiles	Reloads all of the plotfiles in the Plotfiles list.
Reload Selected Plotfiles	Reloads plotfiles highlighted in the Plotfiles list.
Close Selected Plotfiles	Closes plot files highlighted in the Plotfiles list and removes the plotfile from the list.
Close All Plotfiles	Closes all plotfiles displayed in the Plotfile list whether or not they are highlighted and removes the plotfiles from the list.
Close Window	Closes the Signal Manager dialog box.

Signal Manager Plotfile Menu

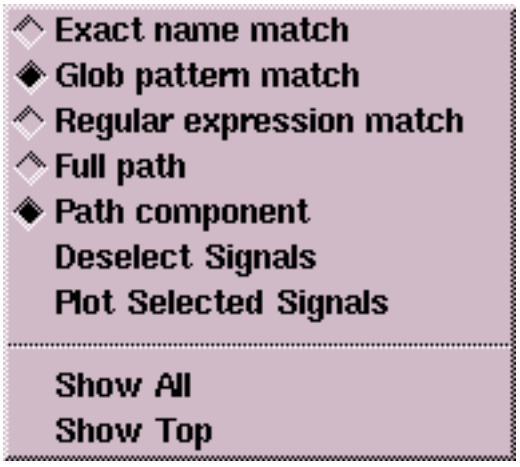
The following items appear in the **Plotfile** menu list in the Signal Manager dialog box:

Display Selected	Opens plotfile windows for the plotfiles highlighted in the Plotfiles list.
Display All	Opens plotfile windows for all plotfiles displayed in the Plotfile list whether or not they are highlighted.

Hide Selected	Closes plotfile windows highlighted in the Plotfiles list. The plotfiles remain listed.
Hide All	Closes all plotfile windows displayed in the Plotfiles list whether or not they are highlighted. The plotfiles remain listed.
Stack Selected	Opens and stacks plotfile windows highlighted in the Plotfiles list. Stack orientation is managed with the Setup dialog box.
Stack Visible	Stacks plotfile windows which are visible on the work surface. Stack orientation is managed with the Setup dialog box.
Stack All	Stacks all plotfile windows in the Plotfiles list whether or not they are highlighted. Stack orientation is managed with the Setup dialog box.

Signal Manager Signals Menu Items

The following items appear in the **Signals** menu list in the Signal Manager dialog box:



Options under Signal Manager - Signals Menu

NOTE

On some Windows systems, the selection buttons do not appear as shown above. Click the left mouse button on the highlighted entry to activate the selection as a check mark.

There are three “Match” options from which to choose. these will be used in conjunction with the Signal Filter field:

- **Exact name match**: will search for an exact name match with your Signal Filter entry;
- **Glob pattern match**: filename pattern matching using *,?[abcd] wildcard characters
- **Regular expression match**: vi/emacs/tcl style patterns using the following wildcard symbols: ^ $. [ ] | ( )

The wildcard characters have the following meanings:

Pattern language	Wildcard Character	Meaning
Glob	*	any sequence of characters
Glob	?	any single character
Glob	[abc]	character a, or b, or c
Regular Expression	.	any character
Regular Expression	<x>|<y>	anything matching regular expression <x>, or regular expression <y>
Regular Expression	( )	parenthesis used for grouping
Regular Expression	$	end of path component, or full path, depending on settings
Regular Expression	^	start of path component, or full path, depending on settings
Regular Expression	[abc]	character a, b, or c
Regular Expression	[^abc]	any character but a, b, or c

Second, select one of two target options:

Full Path, Path Component Specify whether the pattern is matched against single path components, or against full paths. This allows, for example, very specific selections of particular signals for only a given rnage of plotfile containers. The default target for pattern matching is "`path component`."

The last two options on the Signal Manager Signals menu are:

Plot Selected Signals Plots signals that are highlighted in open plot file windows into the active graph window.

Deselect Selected Signals Removes the highlight from highlighted signals in displayed plot file windows.

Signal Manager Signal Filter Field

Enter your filtering pattern in the Signal Filter field; from the adjoining pulldown, select whether the pattern denotes a set to be **Shown**, **Hidden**, **Selected**, or **Deselected**. Choosing the acdtion causes it to be applied to the signals or containers matching the filter pattern in all plot files.

In addition, you may set a **Cumulative** mode. With Cumulative turned off (the default mode), each action on a set of signals or containers denoted by the filter pattern implies that signals or containers not matching the filter pattern have the reverse operation applied to them. For example, if you enter "**v***" as a filter pattern, have "**Show**" as the action, and "**Cumulative**" off, all the v* entries appear, and other entries not matching that pattern disappear. Again, if the action had been "**Select**", all of the v* entries would be highlighted (selected) and other entries not matching that pattern, deselected.

Signal Manager Buttons

The following buttons appear in the Signal Manager dialog box:

Open Plotfiles	Brings up the Open Plotfiles dialog box, allowing you to open new plotfiles. (You can also open plotfiles by using the **File > Open > Plotfile** item in the menu bar or by single clicking on the **Open** icon in the icon bar.)
Close Plotfiles	Closes plotfiles highlighted in the Plotfiles list and removes the plotfile from the list.
Display Plotfiles	Opens plotfile windows for the plotfiles highlighted in the Plotfiles list.
Setup	Opens the Signal Manager Setup dialog box.
Match All	Select this to match corresponding signals in all open plot files to the active graph.
Close	Closes the Signal Manager dialog box.

Signal Manager Setup Button Dialog Box

The **Setup** button opens the Setup dialog box, allowing you to manage stack position, stack orientation, and plotfile window size. A *stack* is an ordered group of plotfile windows, placed one on top of each other.

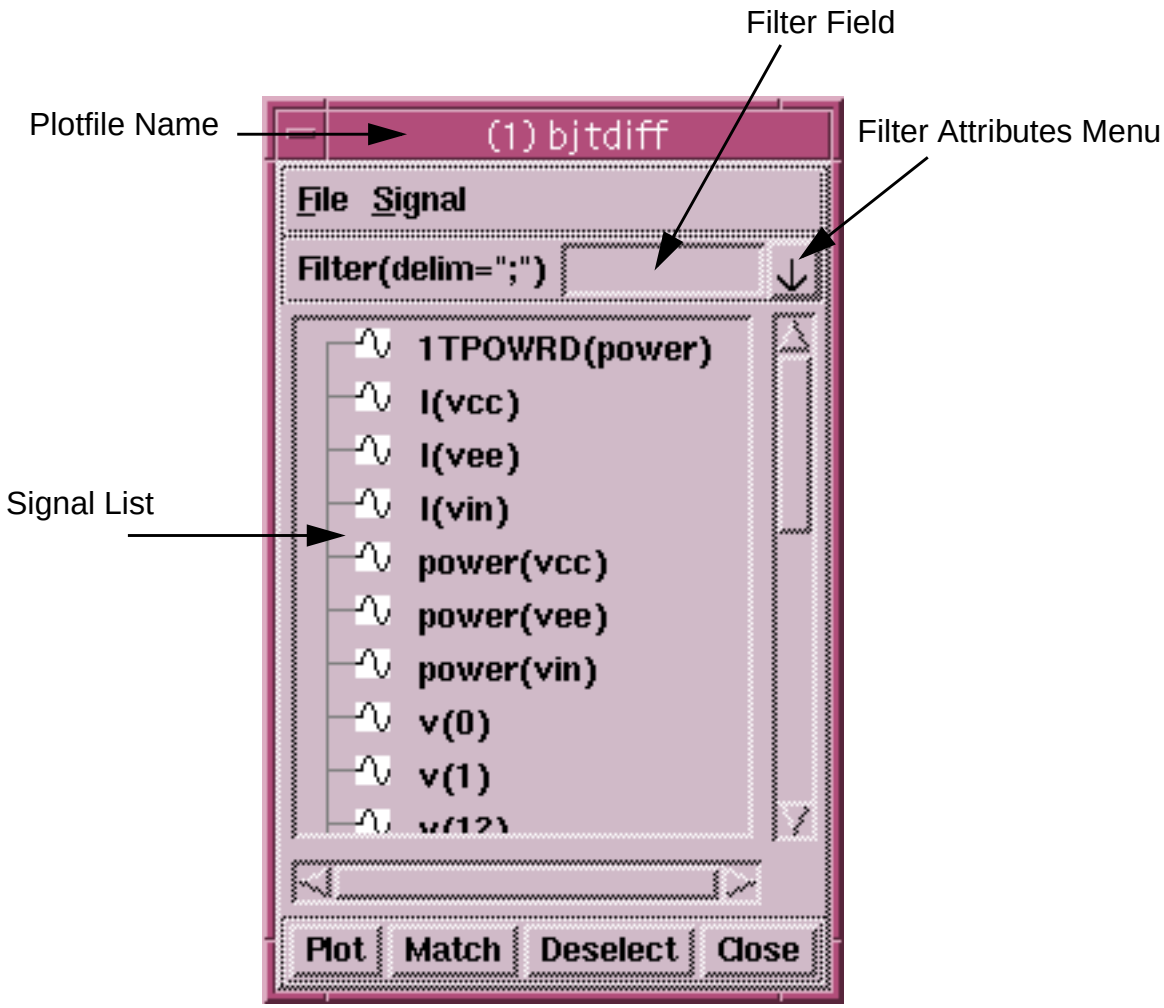
The Setup dialog box is summarized below:

Anchor Position Buttons	Position the stack on your monitor screen. • For example, if you choose **C** (center), the stack is centered in the screen. • If you choose **N**, the stack is offset to the middle and top (“North”) portion of the screen.
Signal Listbox Option Buttons	Change the signal list windows mode. • **Multiple** - Create one signal list window for each opened plot file. • **Signal** - Create one signal list window for all opened plot files to share. Double clicking on the plotfile name in the Signal Manager will set the plotfile active.

Stacking Orientation Buttons	Change the arrangement of plotfile windows in the stack. • **Horizontal** - Stack plotfile windows anchored to the NW, W**,** or SW are arranged from left to right. Windows anchored to the NE, E, or SE are arranged from right to left. • **Vertical -** Stacks windows from top to bottom. • **Diagonal -** Stacks windows diagonally from left to right.
Stack Buttons	Changes in the Setup dialog box are enabled when one of the **Stack** buttons at the right of the dialog box is pressed. • **Stack Visible** - Stacks plotfile windows that are displayed (*i.e.*, that are not iconized) on the work surface. • **Stack Selected** - Opens and stacks plotfile windows from highlighted plotfiles in the dialog box. • **Stack All -** Opens and stacks all plotfiles listed in the dialog box.
Save Button	Saves the settings into the preference file so that the settings will be loaded and applied the next time that CosmosScope is invoked.
Close Button	Closes the Setup dialog box.
Defaults Button	Resets the Setup dialog box to its original settings.
Listbox Width in Lines field	Changes the width of plotfile windows.
Listbox Height in Liness field	Changes the height of plotfile windows.
Stacking Offset In Pixels field	Changes the overlap spacing of the stack.

Signal Manager Plotfile Window

Choosing a plotfile in the Open Plotfiles dialog box opens the Plotfile window.



The Plotfile window displays the name of the plotfile and a scrollable list of signals that can be displayed in a graph window.

The Plotfile window is further described in the following topics:

- Plotfiles Dialog Box Menus
- Plotfiles Dialog Box Fields
- Plotfile Dialog Box Buttons
- Plotfiles Dialog Box Use Notes

Plotfiles Dialog Box Menus

The Plotfile dialog box **File** menu items are summarized as follows:

Reload Plotfile	Reloads selected plotfiles.
Close Plotfile	Closes the plotfile window and removes the file from the Signal Manager Plotfiles list.
Close Window	Closes the plotfile window. The plotfiles remain listed.

The Plotfile dialog box **Signal** menu items are summarized below:

Match options	As in the main Signal Manager window, you may select the signal pattern matching language (exact, glob, or regular expressions).
Path options	Select either **Full** or **Path Component** for the pattern matching target
Deselect Selected Signals	Removes the highlight from highlighted signals in displayed plot file windows.
Plot Selected Signals	Plots signals that are highlighted in open plot file windows into the active graph window.
Show All	Displays all signals in all levels of hierarchy.
Show Top	Displays only those signals at the top level of hierarchy.

Plotfiles Dialog Box Fields

The following list summarizes the Plotfile dialog box fields and buttons, from top to bottom:

Filter(delim="?") Field	This label next to the filter pattern provides the delimiter character used for signal path names for this plotfile. If you wish to match against full path names, separate individual path components with the character ("**?**" in this case). Enter your search pattern, then select the filter attribute from the options in the adjoining pulldown: **Show, Hidden**, **Selected**, or **Deselected**. This attribute is retained when you enter subsequent search patterns, until you reset the selected attribute option. And, as in the main Signal Manager window, you may set or unset the cumulative mode, with the same meaning.
Signal list	The Signal list is a scrollable list of signals that can be displayed in a graph window. Signals are organized by type, such as analog, digital, or event; Verilog designations include bus, register, register_int, and string.
Match Button	Search the current Signal List to find and plot signals that have the same name as the signals in the active graph.
Plot Button	Plots highlighted signals to the active graph window.
Deselect Button	Deselects highlighted signals in the signal list.
Close Button	Closes the plotfile window. The plotfiles remain listed.

Plotfiles Dialog Box Use Notes

Signal Containers

- Within the Plotfile dialog box Signal List , signal containers are always preceded by a **-** or a **+** and contain sets of related signals. A **-** indicates that the signal container is open, and related signals are displayed beneath it. A **+** indicates that the signal container is closed, and no signals are displayed.

  To open a + signal container, double click on the container with the left mouse button.

To select signals for placement into the graph window

- Place the mouse cursor over the signal and single click the left mouse button. Any number of signals can be selected.

To unselect a signal

- Place the mouse cursor over the highlighted signal and single click the left mouse button.

To place the highlighted signals into the active graph window

- Single click on the **Plot** button, or double click on the highlighted signal, or use the middle mouse button to paste the signals into the desired graph region.

To display signal containers or individual signals in the Signal List

- Type the name in the Filter field and press the **Return** key on your work station keyboard.

If you want to display all signals and containers

- Delete all entries in the Filter field and press the **Return** key.

Chapter 6

Graph Window Operation

CosmosScope graphs display data derived from simulation analyses or other sources such as Polaris vcd files. Graph data can be signals, statistics, etc., plotted versus time, frequency or other data.

This chapter explains how to display, open, and, save your graphs, along with graph menu options and dialog boxes. There are two basic kinds of graph regions supported in the graph window: a Trace graph region, where digital signals are displayed, and Analog graph regions;these are also described in this chapter.

The following topics describe the general operations of the Graph windows:

- Displaying a Graph
- Saving a Graph or Outline
- Opening a Saved Graph or Outline
- Redraw Status Window
- Zooming
- Panning
- Scroll Bars
- Slider
- Trace Graph Region
- Analog Graph Region
- Smith Chart
- Polar Chart

Displaying a Graph

To graph waveforms on the work surface, use the Signal Manager.

You may also update a displayed waveform by pressing the Reload button (third from left in the icon bar) or update the waveform dynamically.

To update the waveform dynamically, select **Edit > Graph Preferences**. In the Graph Preferences form, select the Display tab. Turn on Dynamic Waveform Display and enter the interval, in seconds, desired for dynamically updating the displayed waveform.

Saving a Graph or Outline

A `Graph` file contains all of the information displayed in the graph window, including displayed signals and any additions made with the Draw tool or Measurement tool. To save a graph into a file, select the **Save >Graph** menu item from the File menu. This activates the Save Graph dialog box.

The Save Graph dialog box allows you to specify a name. (A directory of that name is created to contain the graph. An empty file is created at the same level, with the same name, but with the extension `.ai_graph` appended to it.) After you click on the **Save** button a pop-up dialog box prompts you to save the graph file in one of the following ways:

1. With a copy of the waveforms in the graph.
2. With a reference to the plot file from which the waveforms in the graph were plotted.

In the first case, copies of the related plot files are saved in the directory that you specified; these plotfiles are now separate from the originals and are not overwritten by subsequent analyses. In the second case, the connection to the original plot files is maintained; thus, if the graph is reopened it can be automatically updated due to any Replace or Append plot actions specified for an analysis. In this case you can select to either save the absolute path or the relative path in the saved graph by checking the appropriate radio button.

An `Outline` file contains all of the information displayed in the document window except for displayed signals. Any additions made with the Draw tool or Measurement tool are included in an outline file. To save a graph outline into a file, select the **Save Outline ...** menu item from the File menu. This activates the Save Outline dialog box.

The Save Outline dialog box allows you to specify a path and file name for the outline. (The extension `.ol` is automatically appended to the filename.) After you click on the **Save** button, the Graph Outline pop-up dialog box allows you to specify several attributes to be saved with the outline. Click on the appropriate checkbutton to save an attribute. Notice that you can select whether or not to maintain the connection to the plot file in the same way as for a graph outline. You control this by checking (or unchecking) the **Dependencies** checkbutton on the Graph Outline dialog box.

Opening a Saved Graph or Outline

To open a saved graph:

- Select the **Open Graph ...** menu item from the File menu. This activates the Open Graphs dialog box.
- Navigate to the directory where the graph file is located.
- Select the file, and click on the Open button. A new Graph window opens and the waveforms are displayed.

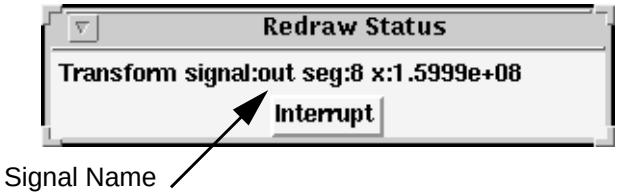
To open a saved outline:

- Select the **Save Outline ...** menu item from the File menu. This activates the Open Outline dialog box.
- Navigate to the directory where the outline file is located.
- Select the file, and click on the Open button. A Create Graph With Outline dialog box opens with a list of waveform placeholders. You must paste waveforms into the list by selecting them in a graph or Plot File window, placing the cursor in the entry box next to the placeholder, and clicking the middle mouse button. You cannot type in the entry boxes.
- Click on **Apply** to open a new Graph window and plot the waveforms.

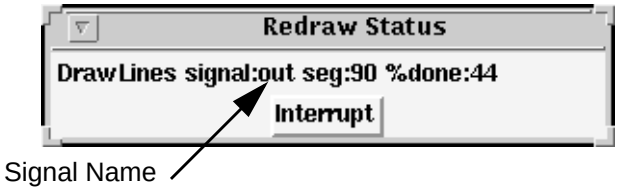
Redraw Status Window

Rendering a large, complex graph after an open, reload, or zoom action can take several moments. The CosmosScope Redraw Status window informs you about the amount of time it will take to redraw the graph window. The intervals at which this is updated are defined in the Graph Preferences dialog form. There are two steps to the redraw process.

The Transform Signal step displays the transformation of data to graphics. Redrawing of the graphic can be halted at any time by single clicking on the **Interrupt** button. Only the percentage of the signal processed before the **Interrupt** button was pressed will be displayed.



The Draw Lines portion displays the percentage of graphics data being written to the graphics window. Redrawing of the graphic can be halted at any time by single clicking on the **Interrupt** button. Only the percentage of the signal processed before the **Interrupt** button was pressed will be displayed.



Zooming

Zooming In

Zooming in on a graph region increases magnification to show increased detail.

There are several ways to zoom in on a graph:

- Use the **Graph > Zoom In** item in the Pulldown menu bar to zoom in on the center of a graph.
- Single click on the **Zoom In** icon in the Icon bar to zoom in on the center of a graph.
- Use the **Zoom In** item from the **Graph** popup menu to zoom in on the center of a graph.
- Place the mouse cursor on the corner of a section that you would like to zoom in on. Press and hold the left mouse button, and move the mouse so that a frame is created around the area you wish to enlarge. Release the mouse button to zoom in.
- Place the mouse cursor at one end of the range you wish to zoom in on either the X or Y axis. Press and hold the left mouse button and move the cursor to the other end of the range you wish to zoom in on. Release the mouse button to zoom in to zoom in on the selected axis.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Range > Zoom In** item, and release the mouse button.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Attributes > Zoom In** icon, and release the mouse button.

Zooming Out

Zooming out on a graph decreases magnification to show less detail, but more of the graphed information.

There are several ways to zoom out on a graph:

- Use the **Graph > Zoom Out** item in the Pulldown menu bar to zoom out on the center of a graph.

- Single click on the **Zoom Out** icon in the Icon bar to his zoom out on the center of a graph.
- Use the **Zoom Out** item from the **Graph** popup menu to zoom out on the center of a graph.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Range > Zoom Out** item, and release the mouse button.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Attributes > Zoom Out** icon, and release the mouse button.

Zooming to Fit

Zooming to fit displays the maximum number of data points to show the entire range of a signal.

There are several ways to zoom to fit:

- Use the **Graph > Zoom to Fit** item in the Pulldown menu bar.
- Single click on the **Zoom to Fit** icon in the Icon bar.
- Use the **Zoom to Fit** item in the **Graph** popup menu.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Range > Zoom to Fit** item, and release the mouse button.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Attributes > Zoom to Fit** icon, and release the mouse button.

Panning

Panning allows you to view parts of the data that are not visible on the current graph.

There are several ways to pan:

- Left click on either the X or the Y axis. Use the **Graph > Selected Axes > Range > Pan Right/Left/Up/Down** items in the Pulldown menu bar.
- Place the mouse cursor on either the X or Y axis. Press and hold the middle mouse button and move the cursor. The axis will pan with the mouse cursor. Release the button and the signal will snap to the new axis coordinates.
- Place the mouse cursor inside of a graph region. Press and hold the middle mouse button, and move the mouse so that the axis scales move with it. Release the mouse button, and the waveform will snap to the new axis coordinates.
- Left click on either the X or Y axis. Use the **Selected Axes > Range > Pan Right/Left/Up/Down** items in the **Graph** popup menu.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Range > Pan Right/Left/Up/Down** items, and release the mouse button.
- Place the mouse cursor over one of the axes or over an axis name in the legend. Press and hold the right mouse button. The **Axis** menu is displayed. Move the mouse cursor to the **Attributes > Pan Right/Left/Up/Down** items, and release the mouse button.
- Press and hold the right mouse button on the X-axis scale. The **Axis** menu is displayed. Move the mouse cursor to the **Attributes** item. Use the Slider bar in the Range field to pan the view.

Scroll Bars

Scroll bars are automatically added to a graph window when the contents of the window cannot be completely displayed.

Slider

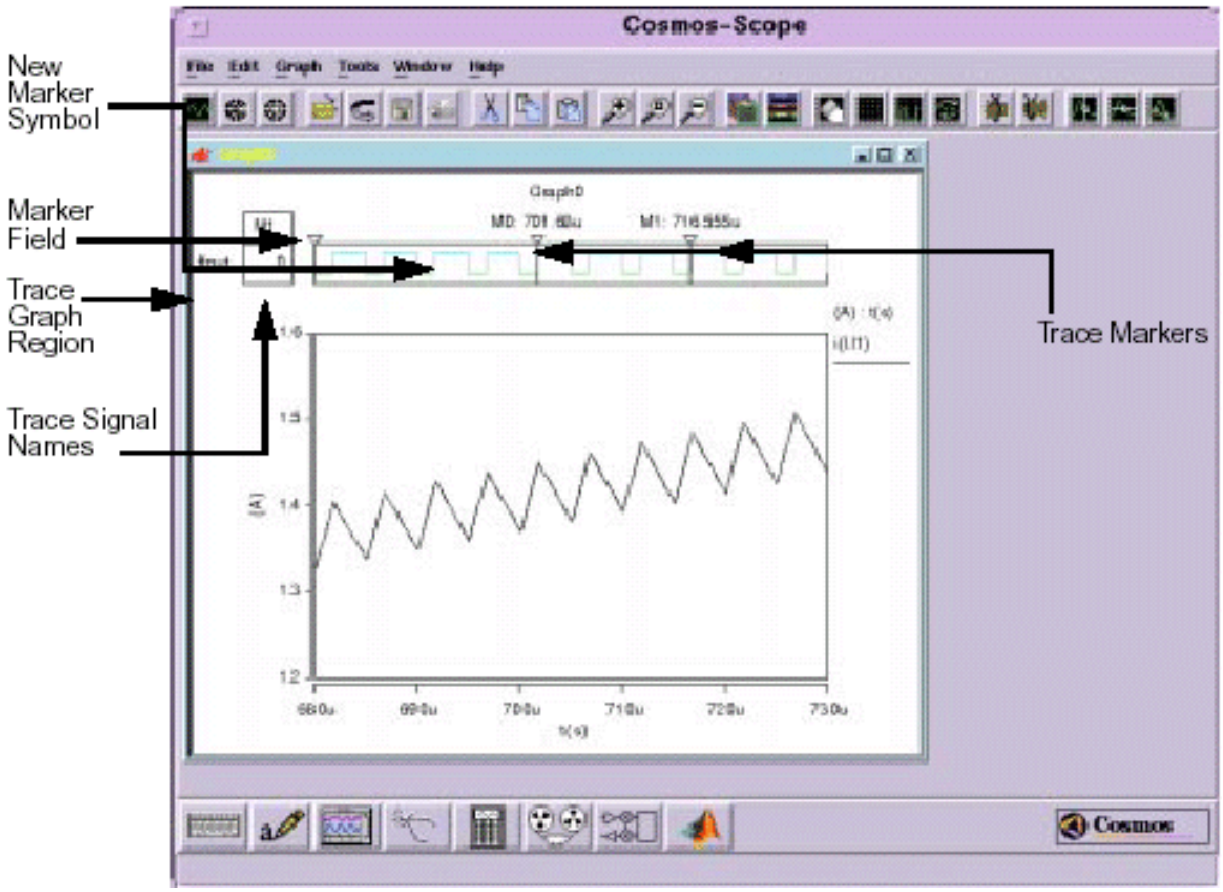
You have the option of using a Slider bar to pan along the X-axis of the graph.

There are several ways to display the Slider bar:

- Left click on the X-axis. Use the **Graph > Selected Axes > Sliders > Show** items in the Pulldown menu bar.
- Left click on the X-axis. Use the **Selected Axes > Sliders > Show** items in the **Graph** popup menu.
- Press and hold the right mouse button on the X-axis scale. The **Axis** menu is displayed. Move the mouse cursor to the **Display Range Slider** item, and release the mouse button.
- Remove the Slider bar from the graph with the **Slider(s) > Hide** item.

Trace Graph Region

The Trace graph region is the primary area where digital signals are displayed. Analog signals can also be placed in the Trace graph region.



The Marker field
Displays the logic state of a point on the digital signal, or the value of an analog signal in the Trace graph region. The point is selected with a Trace marker.

Trace Markers

- To place a Trace marker, highlight the New Marker Symbol with the mouse cursor, press and hold the left mouse button, and drag the marker to the desired location.
- Release the mouse button to place the marker. The logic state of the location is displayed in the Marker field.

The Trace graph uses the same popup menus available in the Analog graph region with the addition of the **Trace** popup menu. The **Trace** popup menu operates only in the Trace graph region.

Analog Graph Region

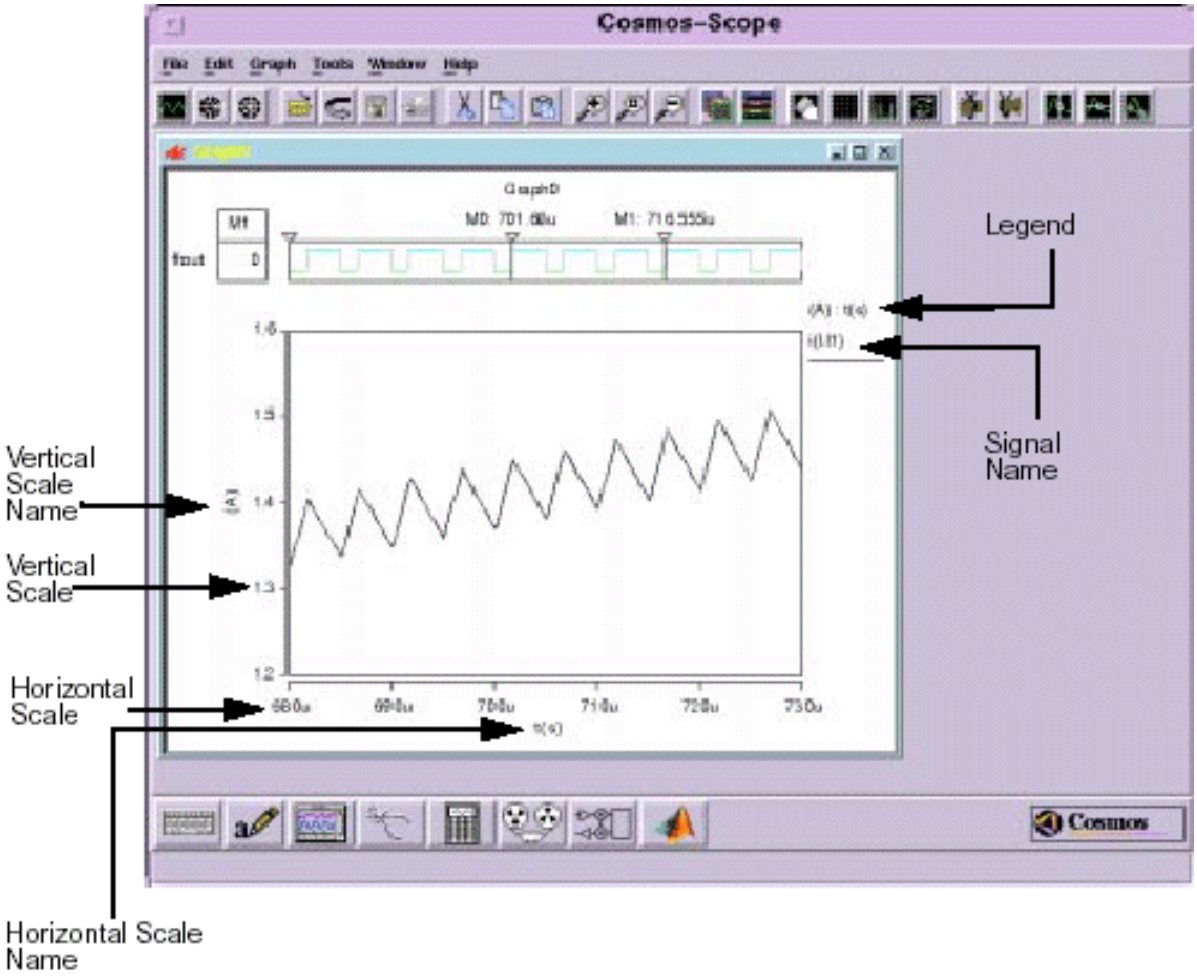
The Analog graph region is the primary area where analog signals are displayed. Digital signals can also be placed in the Analog graph region.

The vertical scale is the Y-axis of the graph. The label for the vertical scale is displayed directly adjacent to the scale and also in the legend displayed next to the graph.

The horizontal scale is the X-axis of the graph. The label for the horizontal scale is displayed directly adjacent to the scale and also in the legend displayed next to the graph.

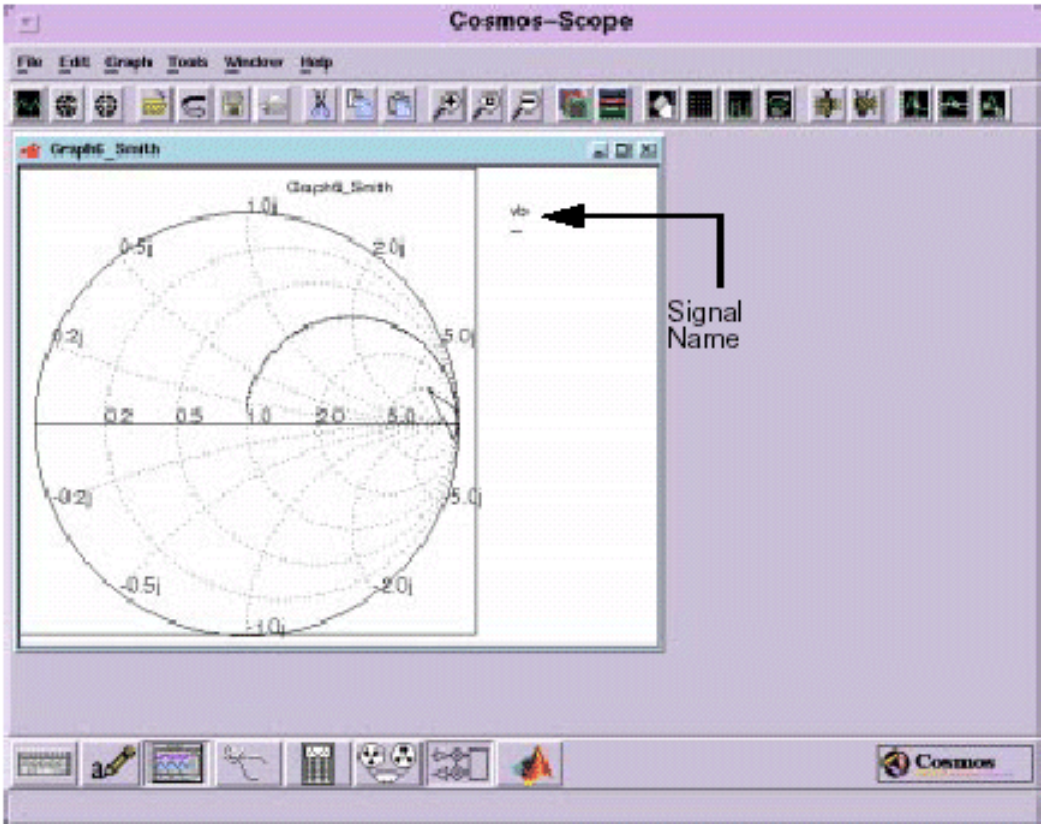
The legend contains the labels for the vertical scale and the horizontal scale, in that order.

The Trace popup menu does not operate in the Analog graph region.



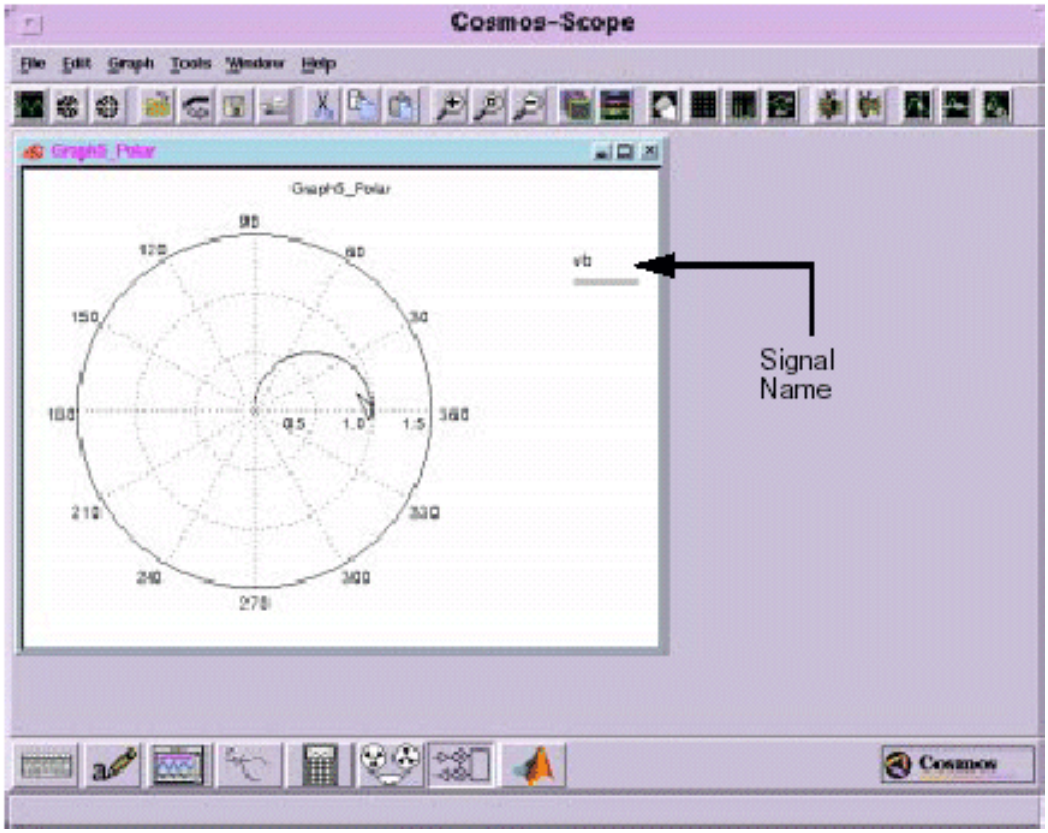
Smith Chart

Smith Chart displays appears as shown below:



Polar Chart

Below you see a Polar Chart dialogue:



Chapter 7

Measurement Tool

The Measurement Tool allows you to perform a variety of measurement operations on displayed waveforms in the CosmosScope Waveform Analyzer. The results of the measurements are displayed in the graph along with the waveform.

If the InSpecs Analysis option is installed, you can use the Measurement Tool with multi-member waveforms.

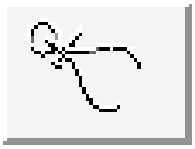
The Measurement Tool is similar to the functions available with the Saber Simulator through its command-line interface or via Saber Guide's **Analysis > Batch** menu option, although some measurements may differ slightly.

The Measurement Tool description is divided into the following topics:

- Accessing the Measurement Tool
- List of Measurement Operations
- Other Measurement Options
- How to Use the Measurement Tool
- Managing Measurement Results
- Multi-Member Waveform Measurements
- Setting Measurement Preferences
- Topline/Baseline Calculation
- Waveform Reference Levels

Accessing the Measurement Tool

The Measurement Tool icon is located in the Tool bar at the bottom of the work surface.



To open or close the Measurement Tool, single click on the icon with the left mouse button.

List of Measurement Operations

The following list identifies the measurements that are available through the Measurement Tool:

Measurement	Category	Description
AC Coupled RMS	Levels	Displays the RMS value of the AC component of a waveform.
Amplitude	Levels	Displays the amplitude of a waveform.
At X	General	Displays the Y-axis value at a particular X-axis point on a waveform.
Average	Levels	Displays the average value of a waveform.
Bandpwidth	Frequency Domain	Displays the bandwidth, the low, high, or center frequency, or the level at which the measurement is made for a bandpass-shaped waveform.
Baseline	Levels	Displays the baseline level of a waveform.
Cpk	Statistics	Displays an indicator of the process capability for a waveform.
Crossing	General	Displays the crossing (intersection) points of two waveforms.
Damping Ratio	s Domain	Displays the damping ratio of a point on a waveform.

Measurement	Category	Description
dB	Frequency Domain	Displays the dB value on a point on a waveform.
Delay	Time Domain	Displays the delay between the edges of two waveforms.
Delta X	General	Displays the X-value difference between two Y-axis points on one or two waveforms.
Delta Y	General	Displays the Y-value difference between two X-axis points on one or two waveforms.
Dpu	Statistics	Displays the number of defects per unit of a scatter plot waveform.
Duty Cycle	Time Domain	Displays the duty cycle of a periodic waveform.
Eye Diagram	Time Domain	Displays the behavior of a waveform cycle during a specific period of time.
Eye Mask	Time Domain	Display the maximum width and height of an eye diagram opening.
Falltime	Time Domain	Displays the falltime of a waveform.
Frequency	Time Domain	Displays the frequency of a periodic waveform.
Gain Margin	Frequency Domain	Displays the gain margin in dB of a complex waveform.
Highpass	Frequency Domain	Displays the corner frequency of a waveform with a highpass shape.
Histogram	Statistics	Displays a histogram of a waveform.
Horizontal Level	General	Displays a moveable horizontal line to identify Y-axis levels.
Imaginary	Frequency Domain	Displays the imaginary value of a point on a waveform.
IP2	RF	Display Input/Output Second Order Intercept Point (I/O IP2)
IP3/SFDR	RF	Display Input/Output Third Order Intercept Point (I/O IP3) or Spurious-Free Dynamic Range (SFDR).

Measurement	Category	Description
Length	General	Displays the length of a straight line that connects two X-axis points on a waveform.
Local Max/Min	General	Displays the local maximum or minimum point on a waveform.
Lowpass	Frequency Domain	Displays the corner frequency of a waveform with a lowpass shape.
Magnitude	Frequency Domain	Displays the magnitude of a point on a waveform.
Maximum	Levels and Statistics	Displays the maximum value of a waveform.
Mean	Statistics	Displays the mean value of a waveform.
Mean +3 std_dev	Statistics	Displays the (mean + 3σ) value of a waveform.
Mean -3 std_dev	Statistics	Displays the (mean -3σ) value of a waveform.
Median	Statistics	Displays the median value of a waveform.
Minimum	Levels and Statistics	Displays the minimum value of a waveform.
Natural Frequency	s Domain	Displays the natural frequency of a point on a waveform.
Nyquist Plot Frequency	Frequency Domain	Displays the frequency at a point on a Nyquist (or Nichols) plot.
Overshoot	Time Domain	Displays the overshoot of a waveform relative to a default or specified topline.
P1db	RF	Display 1DB Compression Point (CP).
Pareto	Statistics	Displays a Pareto chart of a multi-member analysis.
Peak to Peak	Levels	Displays the waveform's peak-to-peak value.
Period	Time Domain	Displays the period of a periodic waveform.
Phase	Frequency Domain	Displays the phase value on a point on a waveform.

Measurement	Category	Description
Phase Margin	Frequency Domain	Displays the phase margin of a complex waveform in degrees or radians.
Point Marker	General	Displays a moveable point marker on the waveform to display the X-value and Y-value.
Point to Point		Displays the Slope value between two selected points.
Pulse Width	Time Domain	Displays the pulse width of a waveform.
Quality Factor	s Domain	Displays the quality factor of a point on a waveform.
Range	Statistics	Displays the range of Y-axis values covered by the waveform.
Real	Frequency Domain	Displays the real value of a point on a waveform.
Risetime	Time Domain	Displays the risetime of a waveform.
RMS	Levels	Displays the RMS value of a waveform.
Settle Time	Time Domain	Displays the settle time of a waveform.
Slew Rate	Time Domain	Displays the slew rate of a waveform.
Slope	Frequency Domain and General	Displays the slope (optionally as a per-octave or per-decade value) of a waveform.
Standard Deviation	Statistics	Displays the standard deviation of a waveform.
Stopband	Frequency Domain	Displays the stopband, the low, high, or center frequency, or the level at which the measurement is made for a stopband-shaped waveform.
Threshold (at Y)	General	Displays the X-axis values at a particular Y-value on the waveform.
Topline	Levels	Displays the topline level of a waveform.
Undershoot	Time Domain	Displays the undershoot of a waveform.

Measurement	Category	Description
Vertical Level	General	Displays a moveable vertical line to identify X-axis levels.
Vertical Cursor	General	Displays a vertical cursor that spans different graphs, for X-value, Y-value, and delta Y measurements.
X at Maximum	Levels	Displays the X-value corresponding to the maximum value of a waveform.
X at Minimum	Levels	Displays the X-value corresponding to the minimum value of a waveform.
Yield	Statistics	Displays the ratio of data points that fall between the specified upper and lower Y-axis values of a waveform.

How to Use the Measurement Tool

To use the measurement Tool you must have waveforms displayed in the graph window.

From the Measurement dialog box you can select the measurement operation you want to perform, select the signal on which to perform the measurement, and select measurement preferences.

After performing measurements with the Measurement Tool, you can manage your measurement results with the Measure Results dialog box.

The following topics describe how to use the Measurement Tool:

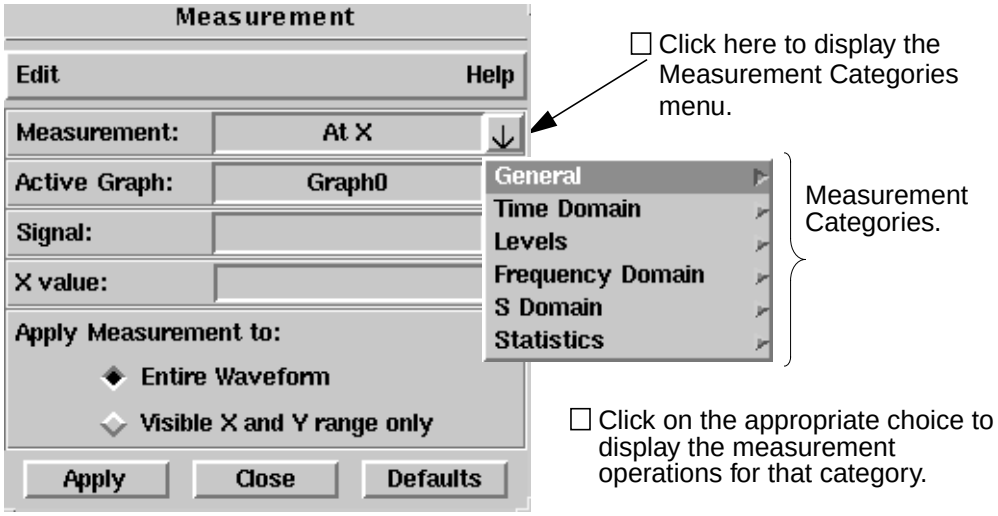
- Measurement Dialog Box
- Selecting a Measurement
- Selecting a Signal for a Measurement
- Setting the Range of a Measurement
- Creating a New Waveform of Measurement Results

Measurement Dialog Box

The Measurement, Signal and Apply Measurement to fields are always available in the Measurement dialog box. Other fields within the dialog box change or

appear according to the particular measurement you are performing. These fields are described in the individual measurement operations.

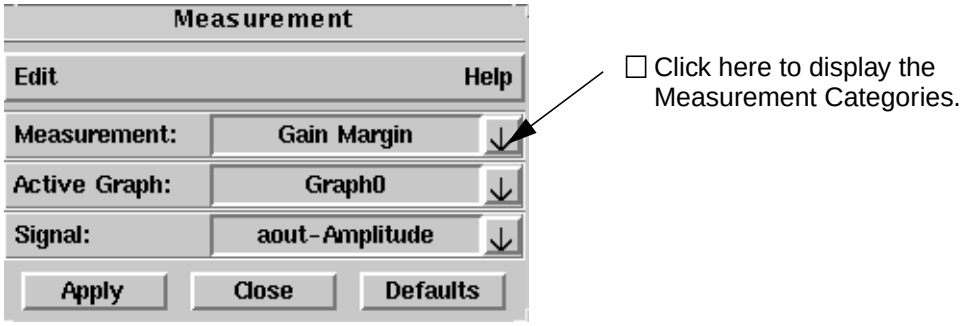
The measurement operations are divided into categories as shown in the Measurement dialog box.



Selecting a Measurement

The Measurement field allows you to select the measurement operation.

- Click on the associated arrow button to expand the list of available measurements and select the one you want.

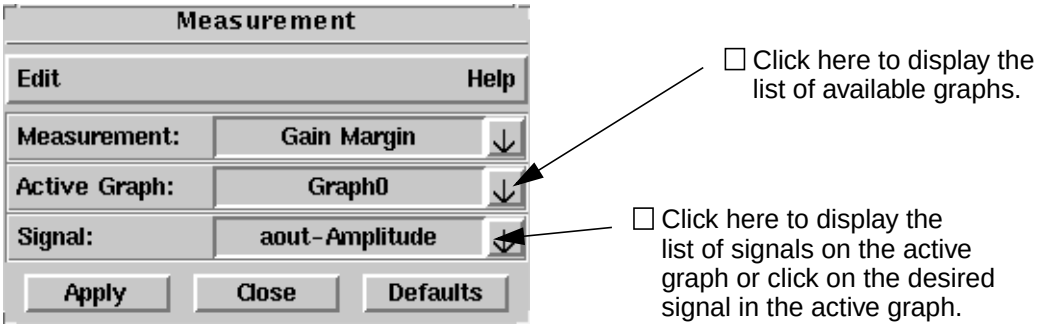


Selecting a Signal for a Measurement

The Signal field allows you to select the signal from the active graph on which a measurement will be performed.

- Click on the associated arrow button to expand the list of available signals from the active graph and select the one you want, or you can click on a signal in the active graph.
- Click on the associated arrow button to expand the list of available graphs and select the one you want.

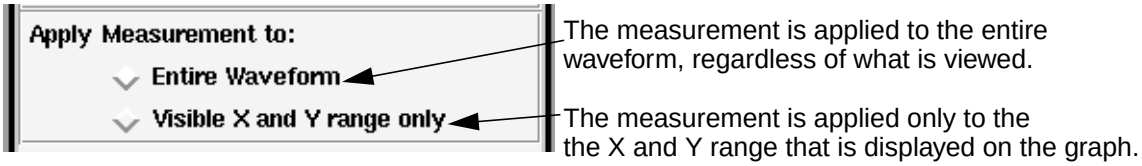
If more than one signal is selected on the active graph, the most recently selected signal is used for the measurement.



Setting the Range of a Measurement

The Apply Measurement to field provides two range choices for most measurements.

- Click on one of the range choices for each measurement performed.

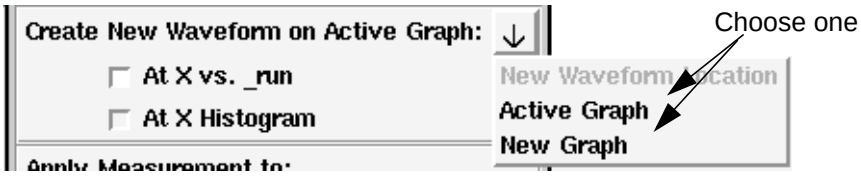


Creating a New Waveform of Measurement Results

The result of some measurements produce other waveforms, especially with multi-member waveforms. The Create New Waveform in Active Graph field allows you the option of creating the new waveform in the active graph or creating a new graph to display the results.

- Click on the downward pointing arrow to display your options.

- Click on Active Graph or New Graph:



NOTE

If any existing waveform is automatically updated due to an automatic plot action (such as Append) specified in an analysis, all measurement waveforms that depend on that waveform are updated at the same time.

Managing Measurement Results

The Measure Results dialog box displays measurement results, and manages the amount of data you view on a graph at one time.

The Measure Results dialog box description is divided into the following topics:

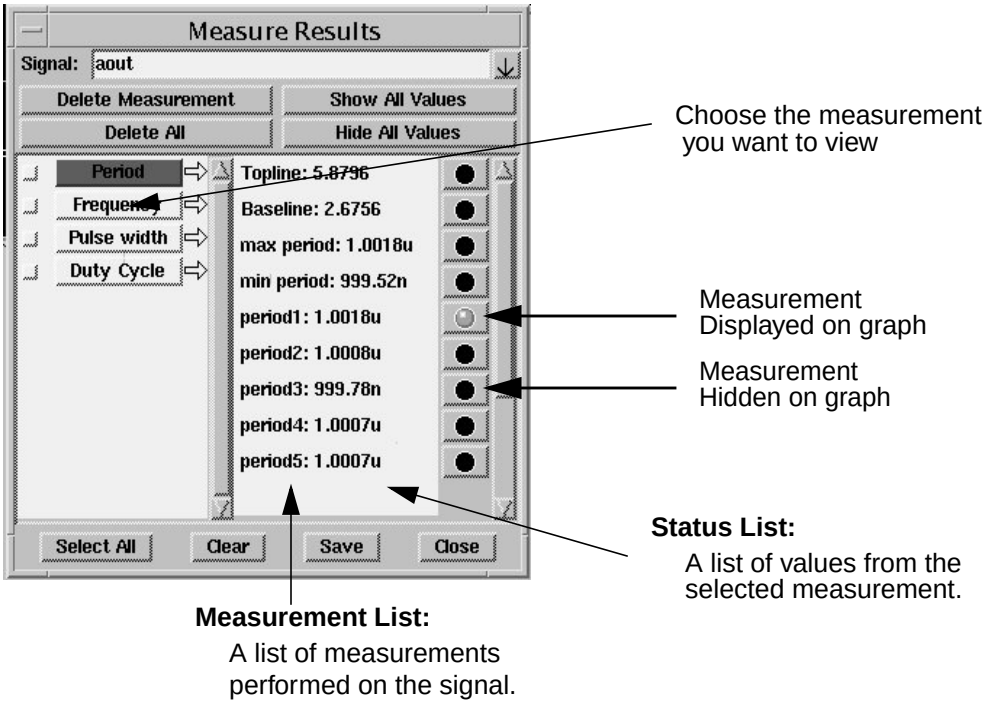
- Accessing the Measurement Results Dialog Box
- Measurement List
- Status List
- Signal Field

Accessing the Measurement Results Dialog Box

There are various ways to open the Measure Results dialog box.

- Select the **Graph > Measure Results** pulldown menu choice.

- Double-click on any highlighted measurement value in a graph window.



The **Delete Measurement** button permanently deletes the selected measurement.

The **Delete All** button permanently deletes all measurements for the selected signal.

The **Show All Values** button displays all values for the selected measurement.

The **Hide All Values** button hides the display of all values for the selected measurement. Passing the mouse cursor over a visibility indicator in the Status list will cause a highlighted measurement value to appear on the signal in the active graph.

The **Select All** button selects all measurements so information for them can be saved to a text file when the “Save” button is selected.

The **Clear** button deselects all the selected check boxes.

The **Save** button pops up a dialog window with information for the selected measurements. The “Save” button is located at the bottom of the window. Clicking this button allows you to save information for the selected measurements to a text file.

The **Close** button closes the Measure Results dialog box.

Measurement List

The Measurement list displays all measurements that are active for a signal. Measurement values for the selected measurement are displayed in the Status list.

Status List

The Status list displays a list of measurement values associated with the selected measurement in the Measurement list. Each value has a visibility indicator associated with it.

A visibility indicator shows the display status of a measurement value.

- To change the state of the visibility indicator, single click the left mouse button.
- An all black visibility indicator indicates that the measurement value is not displayed.
- An all green visibility indicator shows that the measurement value is displayed.

Signal Field

The Signal field allows you to select any signal displayed in the graph window.

- To display all available signals in the graph window, single click with the left mouse button on the downward pointing arrow at the right of the Signal field.

While any signal in the graph window can be selected, the Measure Results dialog box will not show any data for signals that have not been measured with the Measurement Tool.

Multi-Member Waveform Measurements

A multi-member waveform is created as a result of a Vary or Monte Carlo simulation. You must have installed either the InSpecs Parametric Analysis Option or the InSpecs Statistical Analysis Option in order to plot optional multi-member waveforms.

For example, if you are measuring the risetime of a multi-member waveform from a Monte Carlo analysis, you have the choice of measuring the risetime versus the run, the risetime plotted as a histogram, or both.

If you are measuring the risetime of a multi-member waveform from a Vary analysis, you have the choice of measuring the risetime versus the Vary parameter, the risetime plotted as a histogram, or both.

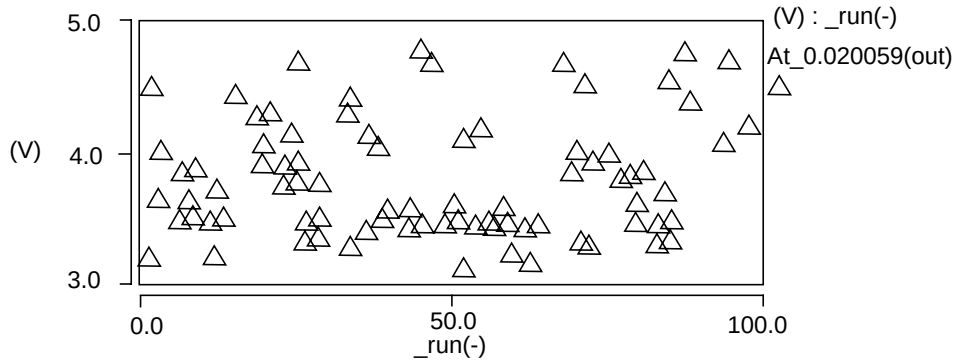
There can be other possible multi-member options provided in the Measurement dialog box depending on what type of measurement you are performing. Each of these options is described for each measurement operation.

Multi-member waveform measurements is further described in the following topics:

- Example of Creating a New Multi-Member Waveform
- Example of Creating a Multi-Member Histogram
- Multi-Member Count
- Multi-Member Count Example

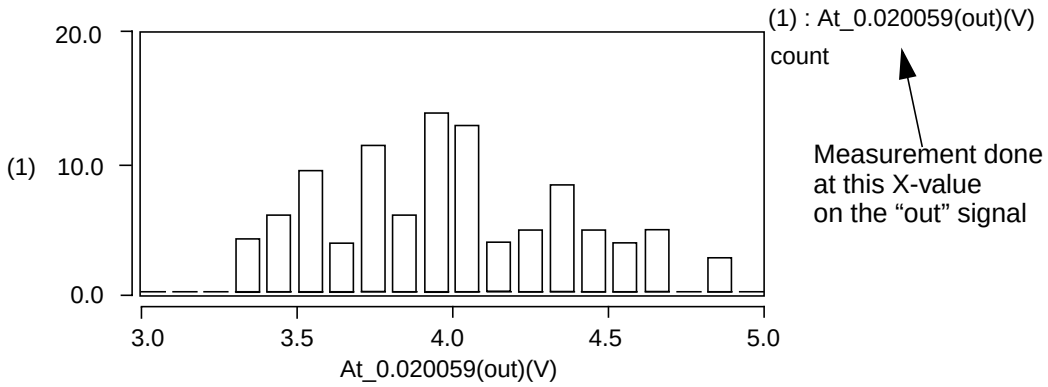
Example of Creating a New Multi-Member Waveform

This scatter plot is generated by performing the **At X** measurement on a Monte Carlo generated, multi-member waveform that contains 100 members. In the Measurement dialog box, you should click on the **At X vs _run** button Each particular Y-value (Y-axis) that occurs at the X-value of 0.020059 volts is plotted against the Monte Carlo run (X-axis) that generates the Y-value.



Example of Creating a Multi-Member Histogram

This histogram is generated by performing the **At X** measurement on a multi-member waveform that contains 100 members. In the Measurement dialog box, you should click on the **At X histogram** button. The number of occurrences (Y-axis) is plotted against the Y-values (X-axis) found at the X-value of 0.020059 volts.

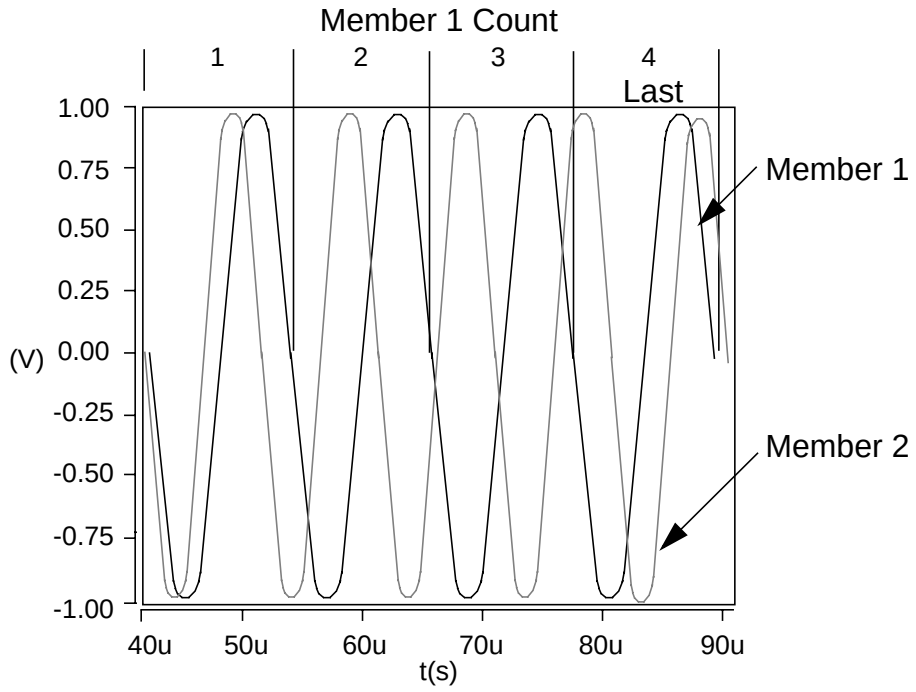


Multi-Member Count

If you have either the InSpecs Parametric Analysis Option or the InSpecs Statistical Analysis Option installed and you are performing a measurement on a multi-member waveform, you can specify a location on a waveform where a measurement is taken. You specify this location in the Measure Preferences dialog box, in the Multi-Member Count field. The waveform location is specified as a count value.

For an example, in the figure below, assume that a frequency measurement is being applied to this waveform. This waveform has two members. Member 1 is shown divided into four counts, each representing periods for this measurement. If a measurement checks the rising edge, then there are four rising edge counts. Although not shown, Member 2 is also divided into counts.

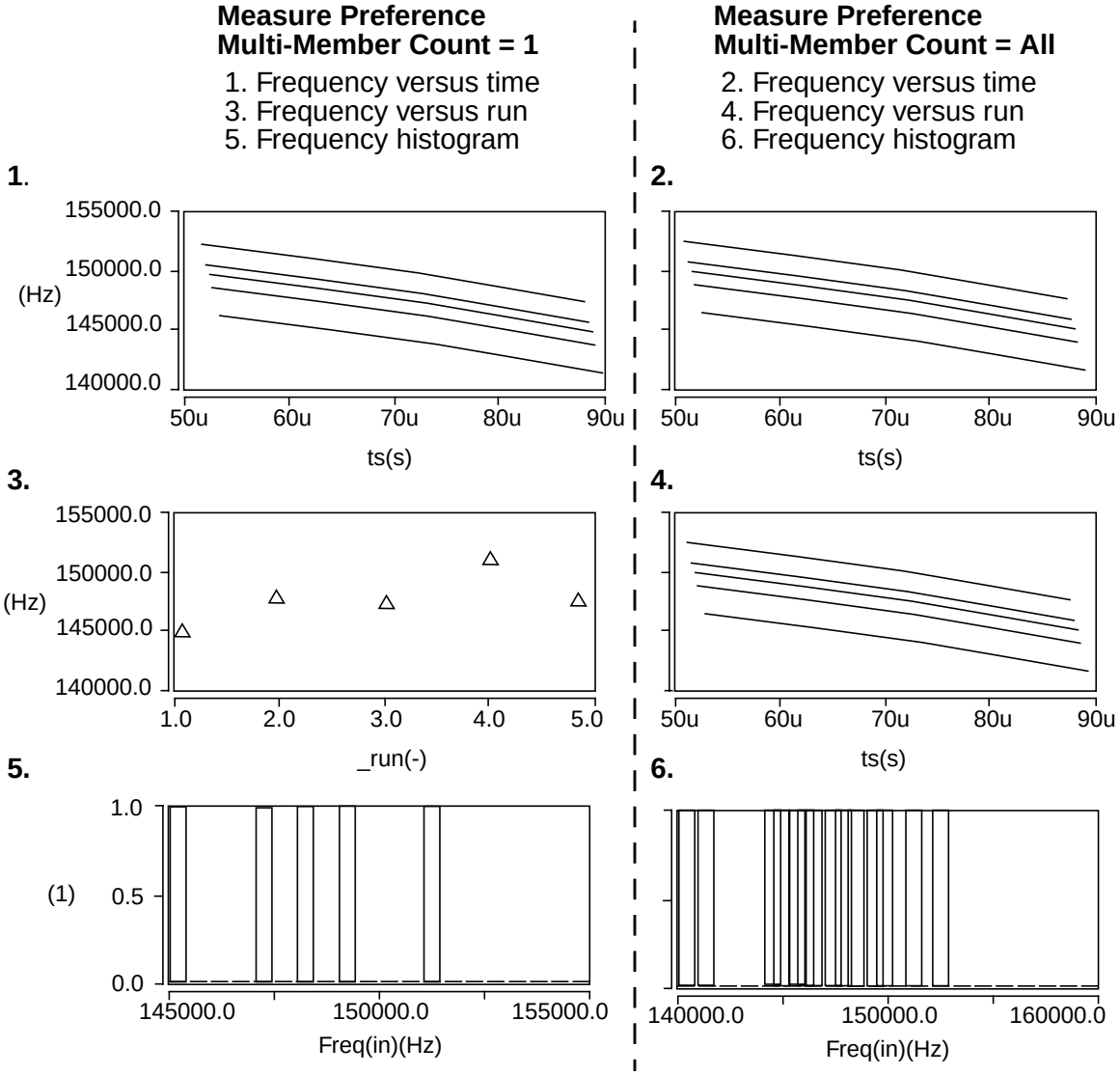
If you use the default count value of 1, the frequency measurement is applied to the first count of each member of the waveform. If you select **Specify** in the Multi-Member Count field, you can choose which waveform count the measurement will use. Specifying **Last** applies the measurement to the last count of each segment. If you specify **All**, the measurement calculation uses all the counts.



Multi-Member Count Example

This topic shows an example of how measurement results vary in response to changing the Multi-Member Count field in the Measure Preference dialog box. This example uses a frequency measurement.

The frequency measurement results shown in the example is for a multi-member waveform with five members, each having eight periods. The frequency of each member in this example is decreased over time. The example shows two measurements, each with three parts: the one on the left was made with the Measure Preferences dialog box, Multi-Member Count field set to **1**. The measurement is made on the first period (or count) of each segment.



Graph 1 shows how the frequency (Y-axis) of each member changes over time (X-axis). Graph 3 is a plot of the frequency (at count = 1) of each of the waveform members plotted against the individual simulation runs (X-axis). Graph 5 shows a histogram, which displays how many occurrences of each frequency value were encountered during all the simulation runs.Graphs 2, 4, and 6 are similar to their counterparts, but they represent all the counts of each waveform member.

Setting Measurement Preferences

Before running a measurement, you can preset Measurement Tool parameters by opening the Measure Preferences dialog box.

You can perform all your measurement operations without ever changing the default values. However, it might be helpful for you to be aware of what these settings are so that you can understand how the results are generated.

To access the Measure Preferences Dialog Box:

- Click on the **Edit > Measure Preferences** menu selection in the Measurement dialog box.

When you change parameters in the Measure Preferences dialog box, click the **Apply** button to cause the change to take affect on the next measurement and the **Save** button to take affect on subsequent invocations of the Measurement Tool.

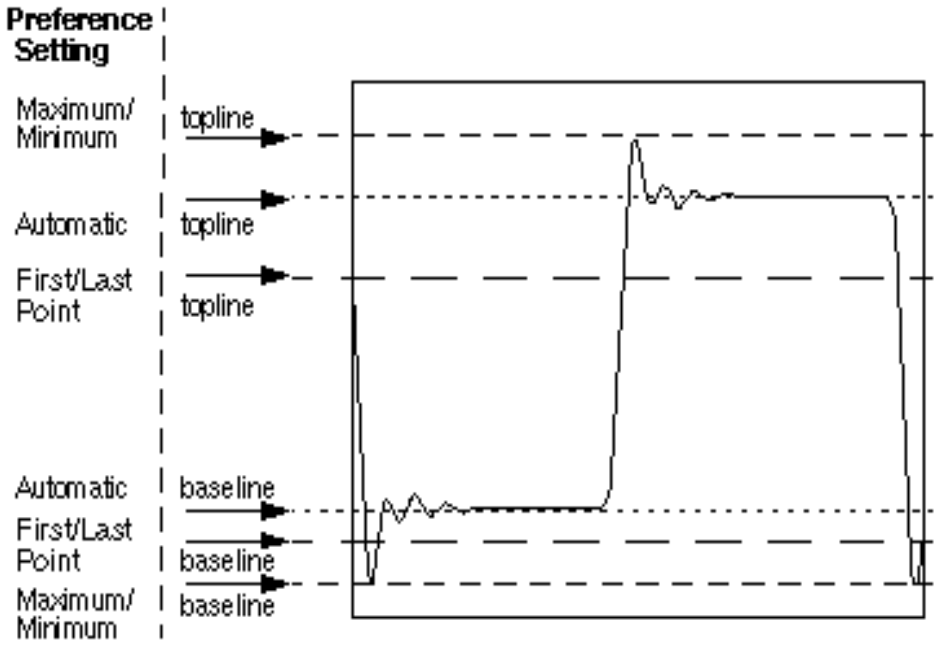
The following table indicates Measure Preferences dialog box options. The **`bold`** choices in the table indicate the default settings.

Parameter	Choices	Description
Display Precision	**5** {a number}	Sets the precision of the numeric results displayed on the graph and in the Measurement Results dialog box.
Default Topline/Baseline	**Automatic**	• When set to **Automatic**, the default is computed by using the method described in Topline/Baseline.
	Maximum/ Minimum	• When set to **Maximum/Minimum**, the maximum and minimum points are used as the topline and baseline values.
	First/Last Point	• When set to **First/Last Point**, the greater level of the first point and last point is used as the topline and the lower level of the first point and last point is used as the baseline.

Parameter	Choices	Description
Multi-Member Count	1	This parameter specifies a location on a waveform where a measurement is taken. This parameter affects waveforms that meet the following criteria:
	All	Operate on a multi-member waveform or two waveforms of separate signals
	Last	Produce more than a single result (such as risetime, period, etc.)
	Specified {a number}	Generate a new waveform.
Histogram Bins	**20** {a number}	For histogram generation, this setting specifies the number of bins displayed. You either use the default of 20 or specify your own integer.
Normalize Histogram	**No** Yes	If yes, the histogram is normalized by dividing the total number of values into each bin count. The result is that each bin is assigned a value between 0 and 1, inclusive.
Cumulative Histogram	**No** Yes	If yes, the histogram is converted to a cumulative histogram by adding the count in each bin to the count of all preceding bins.
Maximum Scattter Plots	Specified {a number}	Specify the maximum number of scatter plots to be placed on the graph. (Note: this applies only to the Pareto measurement.)
Scatter Plots per Graph	1 - 10	Set the Scatter Plots per Graph by clicking the up or down arrow to adjust the selection. (Note: this applies only to the Pareto measurement.)
Sort Result Waveforms	**No** Yes	If yes, the result waveform will be sorted if it is not monotonic.

Topline/Baseline Calculation

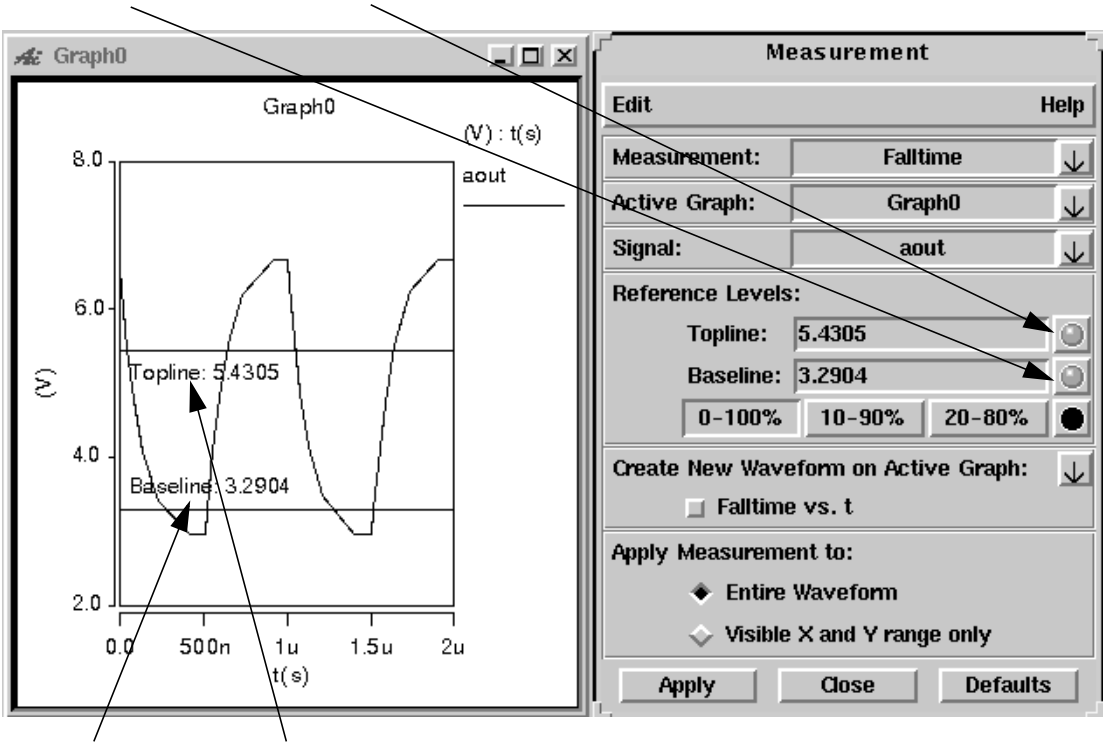
A number of measurements use either specified or default values for topline and baseline in their calculation. You set how your topline and baseline are calculated with the Measure Preferences dialog box. The following figure shows the various topline/baseline possibilities as set in the Measure Preferences dialog box.



Manually Set a Custom Topline/Baseline

There might be times when you want to perform a measurement using a custom top and/or base reference level. The following example shows how to change the topline and baseline references prior to computing a falltime measurement.

1. In the Measurement dialog box, turn on the Baseline and Topline indicator buttons.



Baseline and Topline levels appear in the Graph window.

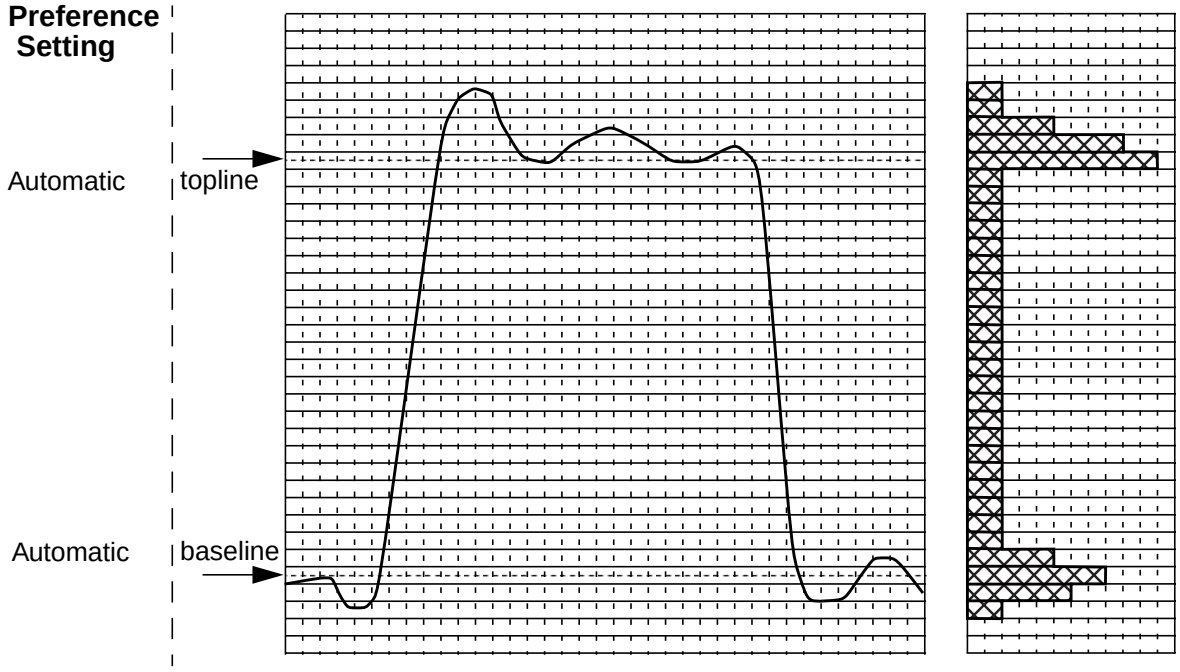
2. Move the cursor to Topline and left-click-and-hold to select it.
3. Move the Topline to the desired value. Repeat for the Baseline.

When using this procedure to change the Topline/Baseline levels, leave the Measurement Reference Level set to the **0-100%** setting so that the measurement is based on the values of Topline and Baseline that you just set.

Default Calculation

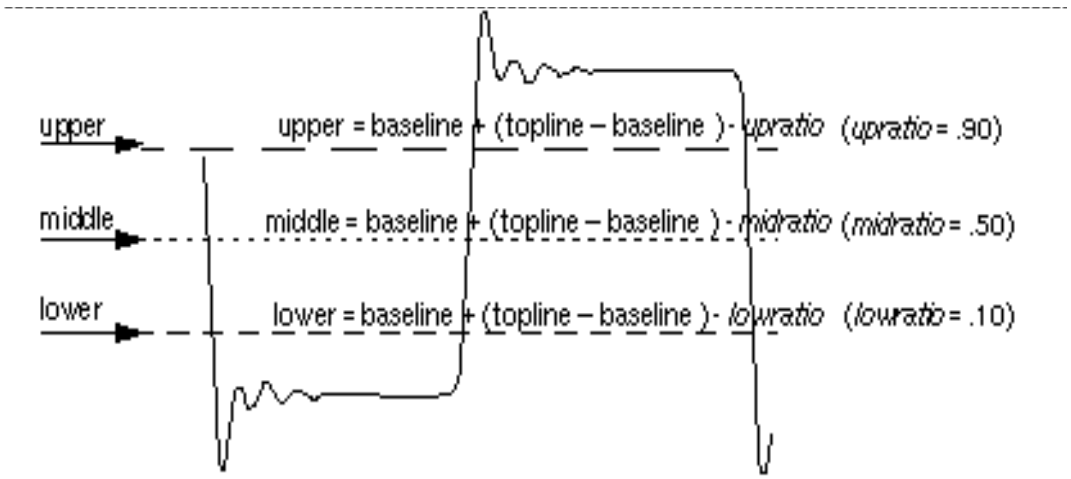
If you use the default setting of **Automatic** in the Measure Preferences dialog box, the baseline and topline levels are calculated by using a probability density histogram method. The waveform is sampled at a number of equally

spaced points along the X-axis. The Y-axis is then equally divided into a number of bins, and the number of sampled points that fall into each bin is determined. The Y-axis bin that contains the most points above the midpoint is used as the topline level. The Y-axis bin that contains the most points below the midpoint is used as the baseline level. This procedure is shown graphically in the following figure with the probability density histogram.



Waveform Reference Levels

In addition to the topline/baseline and maximum/minimum reference levels, other levels are calculated and used by various measurements as shown below.



The reference levels upper, middle, and lower correspond to the distal, mesial, and proximal reference levels defined in the IEEE standard *Pulse Terms and Definitions* (IEEE Std 194-1977).

AC Coupled RMS

Description

Displays the RMS value of the AC component of a waveform.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Category List	All `Levels` category items appear below the Signal field. Select the AC Coupled RMS item and any other items you want to measure.

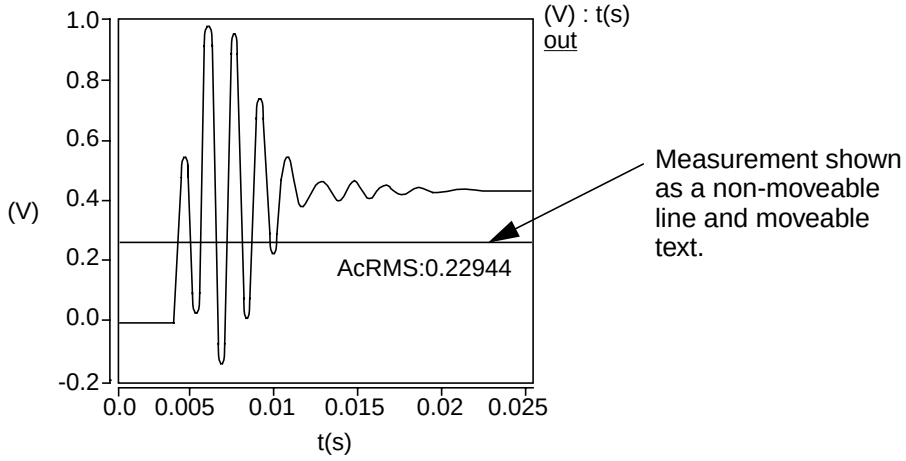
AC Coupled RMS Calculation

The AC coupled RMS value of a waveform is calculated as follows:

$$\left[\frac{1}{(x2-x1)}\int_{x1}^{x2}(W - \overline{W})^2 dx\right]^{1/2}$$

In this calculation, W represents the waveform, $\overline{W}$ is its average value, and x1 and x2 are the starting and ending points for the waveform.

Example



Multi-Member Waveform Options

Create New Waveform on Active Graph or New Graph	Creates a new waveform in the **Active Graph** or in a **New Graph** to display the results. *Measurement* vs. _run or *vary_parameter* - Creates a scatter plot or analog waveform with the current measurement values (Y-axis) versus each Monte Carlo run (X-axis) or Vary parameter value (X-axis). *Measurement* Histogram - Creates a histogram with a count (Y-axis) of the current measurement value (X-axis) occurrences.

Amplitude

Description

Displays the amplitude of a waveform.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog

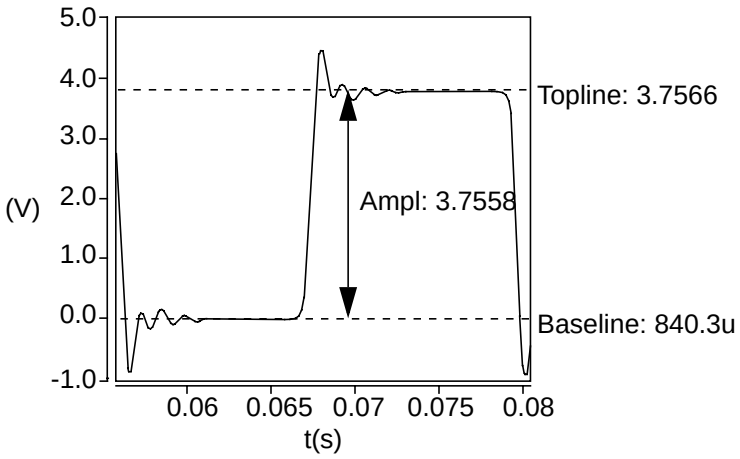
Dialog Box Fields

Category List	All `Levels` category items appear below the Signal field. Select the Amplitude item and any other items you want to measure.

Amplitude Calculation

The amplitude is calculated as the difference between the topline and the baseline reference levels.

Example



At X

Description

Displays the Y-axis value at a particular X-axis point on a waveform.

Measurement Category

General

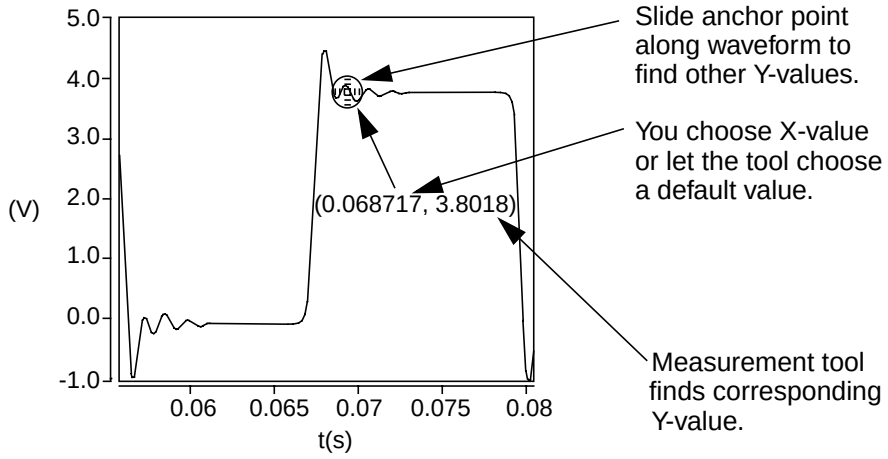
Type of Measured Waveform

Analog, event-driven analog, digital, scatter plot, spectral, histogram, bus

Dialog Box Fields

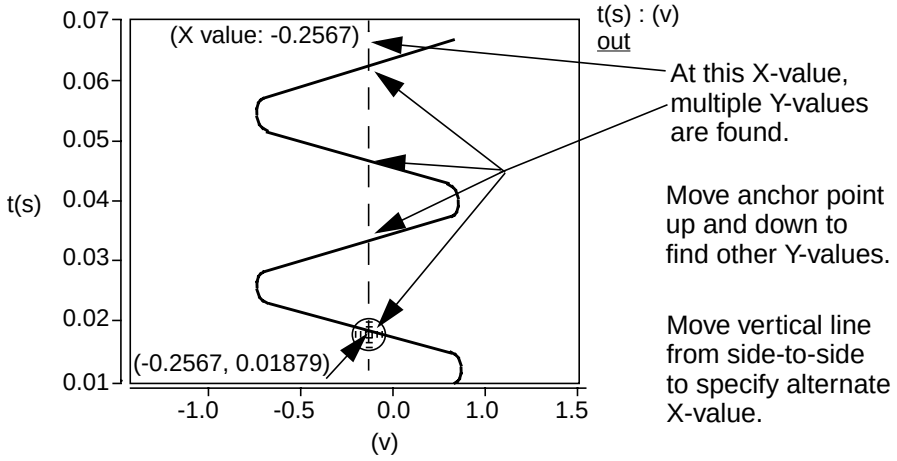
X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

Example 1



Example 2

This example shows a non-monotonic signal. The measurement result can display multiple Y-values for a given X-value with this type of signal.



Average

Description

Displays the average value of a waveform.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Category List	All Levels category items appear below the Signal field. Select the item(s) you want to measure.

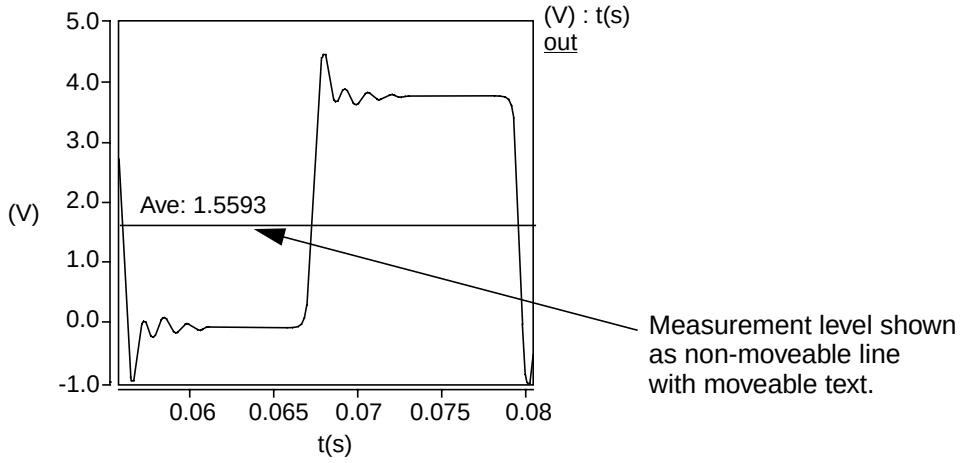
Average Calculation

The average value of a waveform is calculated as follows:

$$\frac{1}{(x2-x1)}\int_{x1}^{x2} W dx$$

W represents the waveform, and x1 and x2 are the starting and ending points for the waveform.

Example



Bandwidth

Description

Displays the bandwidth, the low, high, or center frequency, or the level at which the measurement is made for a bandpass-shaped waveform. The measurement is made relative to a default or specified topline level and a specified offset.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog, spectral

Possible Errors

An error is reported if the first or last data point is above the measurement level.

Dialog Box Fields

Reference Levels

If you want to see the topline and/or offset level displayed on the waveform, you click on the Visibility Indicator to the right of the Topline or Offset field.

Topline
You set this field to a default or a specified level.

Offset
You specify an offset value, to be applied relative to the Topline value. The default is 3. You must also choose which operator to use (–, +, *, or /) along with the specified level. The default is the minus sign. This resulting level is also called the measurement level.

Bandwidth Calculations

The following subtopics describe the calculations to determine the bandwidth of a waveform:

Topline

If you do not specify the topline, a default value is calculated by using a method specified in the Default Topline/Baseline field in the Measurement Preference dialog box.

Offset

Computed as one of

```
topline - offset_value,
topline + offset_value,
topline * offset_value, or
topline / offset_value,
```

The resulting offset (measurement) level is used to determine the bandwidth measurement as follows: (Also see the example.)

fLow (frequency-low) is the first point that falls below the measurement level, before the maximum point. The fLow point can be shown on the graph using the Measure Results dialog box.

fHigh (frequency-high) is the first point that falls below the measurement level after the maximum point. The fHigh point can be shown on the graph using the Measure Results dialog box.

fCenter (frequency-center) is calculated as $\sqrt{fHigh \cdot fLow}$
The fCenter point can be shown on the graph using the Measure Results dialog box.

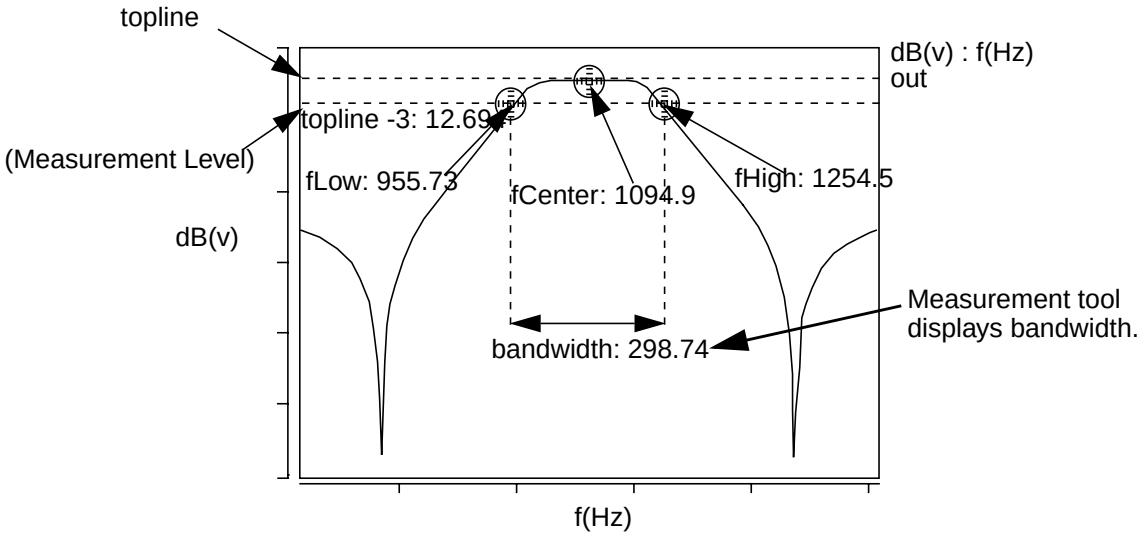
Bandwidth

Calculated as fHigh – fLow.

Q (quality factor)

Calculated by dividing fCenter (frequency-center) by the bandwidth.

Example



Bandwidth Multi-Member Waveform Options

Create New waveform on Active Graph or New Graph

Creates a new waveform in the **Active Graph** or in a **New Graph** to display the results.

Bandwidth vs. _run or *vary_parameter* - Creates a scatter plot or analog waveform with Bandwidth values (Y-axis) versus each Monte Carlo run (X-axis) or Vary parameter value (X-axis).

fLow vs. _run or *vary_parameter* -
Creates a scatter plot or analog waveform with low frequency values (Y-axis) versus each Monte Carlo run (X-axis) or Vary parameter value (X-axis).

fCenter vs. _run or *vary_parameter* -
Creates a scatter plot or analog waveform with center frequency values (Y-axis) versus each Monte Carlo run (X-axis) or Vary parameter value (X-axis).

fHigh vs. _run or *vary_parameter* -
Creates a scatter plot or analog waveform with high frequency values (Y-axis) versus each Monte Carlo run (X-axis) or Vary parameter value (X-axis)

Bandwidth Histogram - Creates a histogram with a count (Y-axis) of bandwidth value (X-axis) occurrences.

fLow Histogram - Creates a histogram with a count (Y-axis) of fLow value (X-axis) occurrences.

fCenter Histogram - Creates a histogram with a count (Y-axis) of fCenter value (X-axis) occurrences.

fHigh Histogram - Creates a histogram with a count (Y-axis) of fHigh value (X-axis) occurrences.

Baseline

Description

Displays the baseline level of a waveform as determined by the Measure Preferences **Default Topline/Baseline** setting.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Category List	All Levels category items appear below the Signal field. Select the Baseline item and any other items you want to measure.

Baseline Calculation

The baseline calculation method is determined by the Default Topline/Baseline field in the Measurement Preference dialog box, which you can set before performing a measurement. If you do not set it, the default method is used.

Cpk

Description

Displays an indicator of the process capability for a waveform relative to specified upper and lower limits.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot

Dialog Box Fields

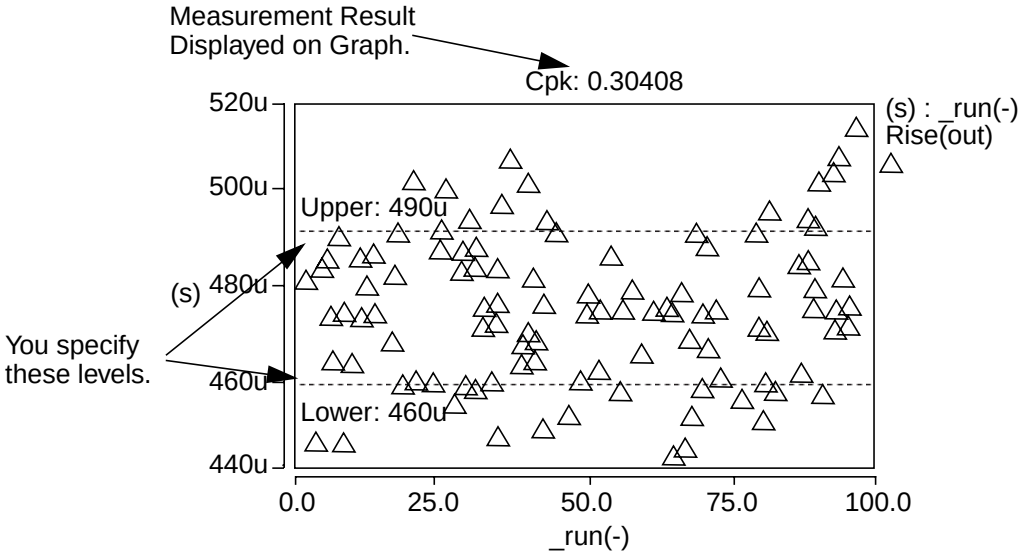
Category List	All Statistic category items appear below the Signal field. Select the Cpk item and any other items you want to measure.
Specification Limit	Required values you provide.
Upper	Specifies the upper specification limit.
Lower	Specifies the lower specification limit.

CPK Calculation

$$\frac{\text{(upper-mean)}}{\text{3 (stddev)}} \text{ or } \frac{\text{(mean-lower)}}{\text{3 (stddev)}}$$

In this calculation, `mean` represents the mean value of the scatter plot, `upper` and `lower` represent the specification limits you specify, and stddev represents the standard deviations of the scatter plot.

Example



Crossing

Description

Displays the crossing (intersection) points of two waveforms.

Measurement Category

General

Type of Measured Waveform

Analog, event-driven analog (The two signals do not need to be the same type.)

Dialog Box Fields

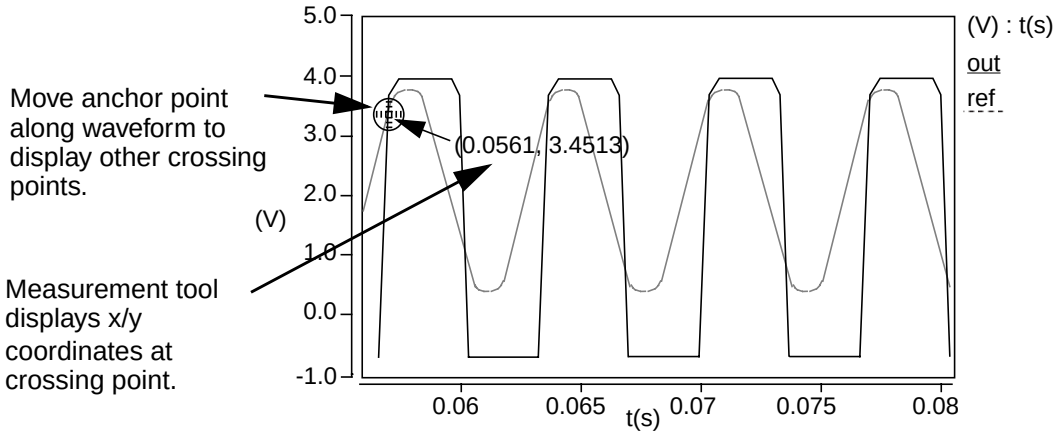
Signal and Ref. Signal	You specify the name of the signal as in all waveform measurements, but in addition, you specify a reference signal. By using the **Swap** button, you can alternate between the two signals to determine which one becomes the reference.
Slope Trigger	either - Measurement finds all crossings regardless of the slopes. positive - Measurement finds crossings where the slope difference between the two waveforms is positive. negative - Measurement finds crossings where the slope difference between the two waveforms is negative.
Create New Waveform on Active Graph or New Graph	X at Crossing vs. *{X-axis}* - Creates a new waveform with the X-values (Y-axis) of the crossing points versus the X-axis parameter. Y at Crossing vs. *{X-axis}* - Creates a new waveform with the Y-values (Y-axis) of the crossing points versus the X-axis parameter as in Example 2.

Crossing Calculation

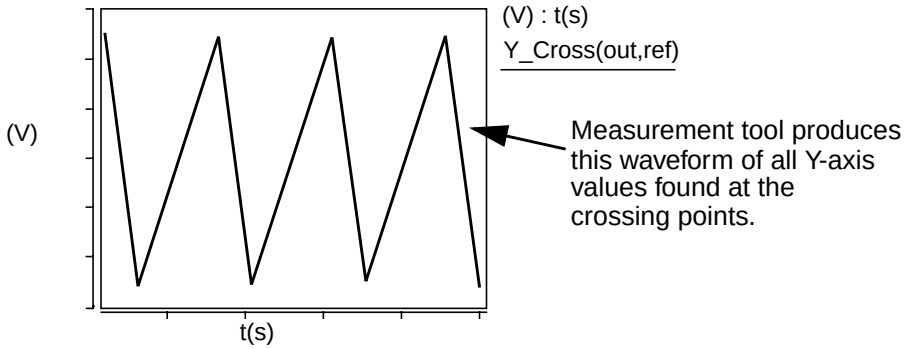
The particular crossing displayed is determined by the Multi-Member Count setting in the Measure Preferences dialog box and by the Slope Trigger field in the Measurement dialog box. If you need more information on the count setting, refer to the Multi-Member Count Example.

The slope of the crossing(s) to be selected can be designated using the Slope Trigger field in the Measurement dialog box. A positive crossing is one where the difference in the slope of the measured waveform and the slope of the reference waveform is greater than 0. A negative crossing is one where the difference of the slopes is less than 0.

Example 1



Example 2



Multi-Member Waveform Options

Create New Waveform on Active Graph or New Graph	X at Crossing vs. _run or *vary_parameter* - A new waveform is computed with X-axis values (Y-axis) versus each run (X-axis) or vary parameter (X-axis). Y at Crossing vs. _run or *vary_parameter* - A new waveform is computed with Y-axis values (Y-axis) versus each run (X-axis) or vary parameter (X-axis). X at Crossing Histogram - A histogram is computed with X-axis values (Y-axis) versus each run (X-axis). Y at Crossing Histogram - A histogram is computed with Y-axis values (Y-axis) versus each run (X-axis).

Damping Ratio

Description

Displays the damping ratio of a point on a waveform.

Measurement Category

s Domain

Type of Measured Waveform

Pole zero data, analog (must be complex)

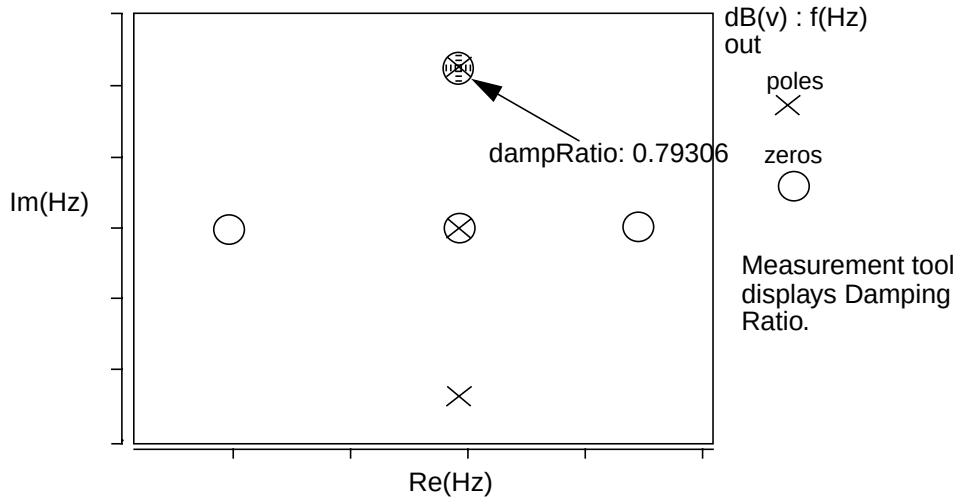
Dialog Box Fields

X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

Damping Ratio Calculation

The damping ratio of a waveform is calculated as -real(magnitude(value)).

Example



dB

Description

Displays the dB value on a point on a waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

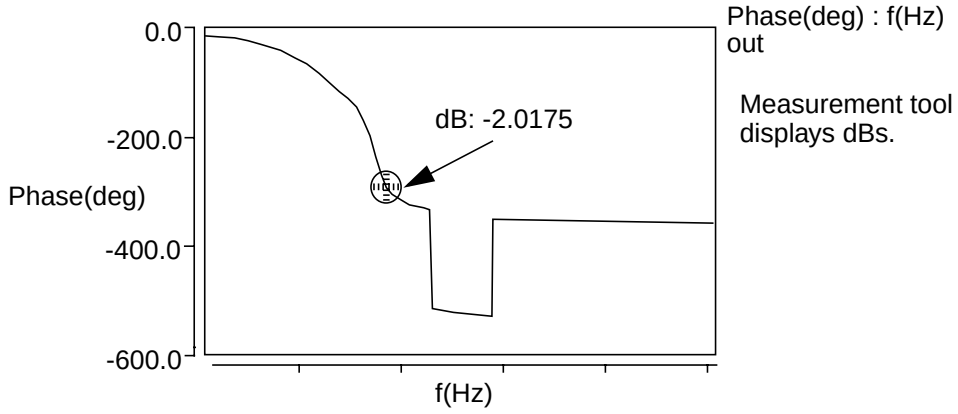
Dialog Box Fields

X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

dB Calculation

The dB of a point is calculated by returning 20(log) of the absolute value of the point on the waveform.

Example



Delay

Description

Displays the delay between the edges of two waveforms relative to default or specified topline and baseline levels for both the measured waveform and the reference waveform. The two waveforms do not need to be the same type, but they must be in the same graph region. It is assumed that the rising or falling edge on the reference waveform causes the corresponding (rising or falling) edge on the measured waveform so that the reference edge occurs before the measured edge.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog, digital

Dialog Box Fields

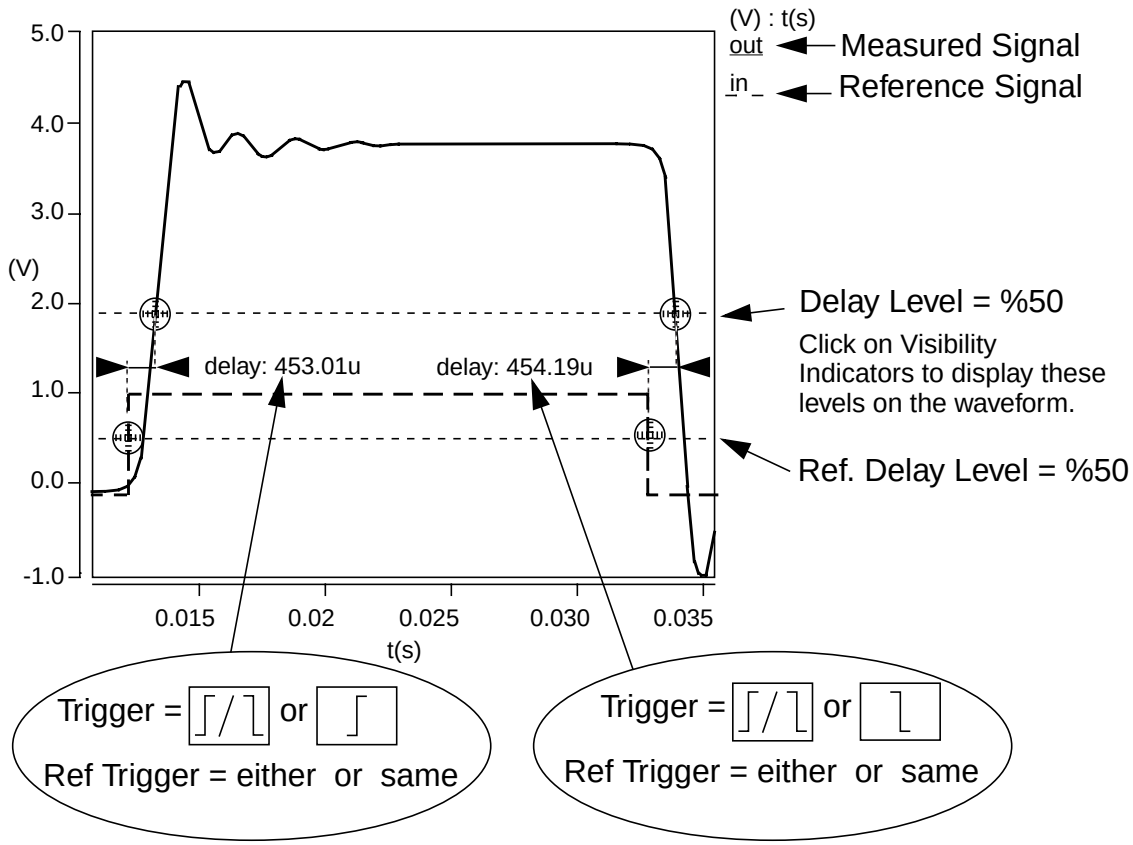
Signal and Ref. Signal	You specify the name of the signal as in all waveform measurements, but, in addition, you specify a reference signal. By using the **Swap** button, you can alternate between the two signals to determine which one becomes the reference.
Reference Levels	The following four fields set the topline and baseline levels for the measured signal and the corresponding reference signal. You can display any of these four levels on the waveform by clicking on the corresponding Visibility Indicator to the right of each field. Topline Specify a topline value or use the default value. Baseline Specify a baseline value below the topline value or use the default value. Ref. Topline Same as Topline, but it applies to the reference signal. Ref. Baseline Same as Baseline, but it applies to the reference signal.

Delay Level	To set the waveform level where the delay is calculated on the measured signal, click on 10%, 50%, or 90%. See the Example.
Ref. Delay Level	Click on 10%, 50%, or 90% to set the waveform level where the delay is calculated on the reference signal. See the Example.
Trigger	 Specifies that the measurement starts from either a rising or falling edge.
	 Specifies that the measurement starts from a rising edge.
	 Specifies that the measurement starts from a falling edge.
Ref. Trigger	either - Selecting this button causes the measurement to trigger on the first edge of the reference waveform prior to the specified measured waveform edge, be it a positive or a negative edge. same - The measurement triggers on the reference waveform at the previous edge with the same polarity (set in the Trigger field) as the measured waveform. opposite - The measurement triggers on the reference waveform at the previous edge with the opposite polarity (set in the Trigger field) as the measured waveform. See the example.
Create New Waveform on Active Graph or New Graph	Delay vs. t - Creates a new waveform with the delay values (Y-axis) versus time (X-axis).

Delay Calculation

All rising or falling edges for the measured waveform are determined based on the Trigger setting in the Measure dialog box. From each edge, the corresponding previously-occurring edge on the reference waveform is determined. The difference on the X-axis between the two edges is the delay time.

Example



Delta X

Description

Displays the X-value difference between two Y-axis points on one or two waveforms. If two waveforms are selected, the two waveforms do not need to be the same type, but they must be in the same graph region.

Measurement Category

General

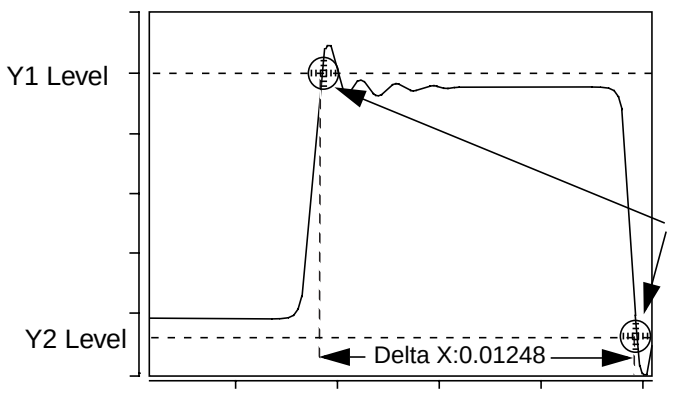
Type of Measured Waveform

Analog, event-driven analog, scatter plot

Dialog Box Fields

Anchor Snap	Signal - Generates moveable anchor points that snap to the signal waveform. See the Example. Closest Measurement - When you have multiple anchor points visible on a graph from previous measurements, this setting causes the measurement to snap to one of those nearest points. If there is no visible measurement on the active graph from which to snap, an error message is displayed. Refer to the Point Marker Example. Floating - Generates moveable anchor points that can be positioned anywhere within the graph. Refer to the Delta Y Example.
Location (Optional)	You can optionally specify two Y-values that are used to determine the X-value difference. See the Example.
Y1 Value	Optional value. You can change it by moving the anchor point after the initial measurement is made.
Y2 Value	Optional value. You can change it by moving the anchor point after the initial measurement is made.
Lock Vertical	An optional check box. If this is checked, the distance of "Y1 Value" and "Y2 Value" will be fixed, so the two anchors of the DeltaX measurement will move together.

Example



Example measure showing anchor points that snap to "Signal."

Slide anchor points along waveform to display other Delta-X values.

Delta Y

Description

Displays the Y-value difference between two X-axis points on one or two waveforms. If two waveforms are selected, the two waveforms do not need to be the same type, but they must be in the same graph region.

Measurement Category

General

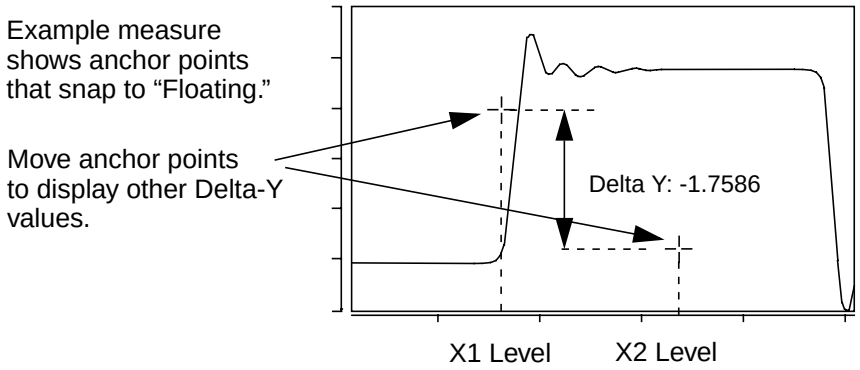
Type of Measured Waveform

Analog, event-driven analog, scatter plot

Dialog Box Fields

Anchor Snap	Signal - Generates moveable anchor points that snap to the signal waveform. To see an example of this type of snap, refer to the Delta X Example. Closest Measurement - When you have multiple anchor points visible on a graph from previous measurements, this setting causes the measurement to snap to one of those nearest points. If there is no visible measurement on the active graph from which to snap, an error message is displayed. Refer to the Point Marker Example. Floating - Generates moveable anchor points that can be positioned anywhere within the graph. See the example.
Location (Optional)	You can optionally specify two X-values that are used to determine the Y-value difference. See the Example.
X1 Value	Optional value. You can change it by moving the anchor point after the initial measurement is made.
X2 Value	Optional value. You can change it by moving the anchor point after the initial measurement is made.

Example



Dpu

Description

Displays the number of defects per unit of a scatter plot waveform.

Measurement Category

Statistics

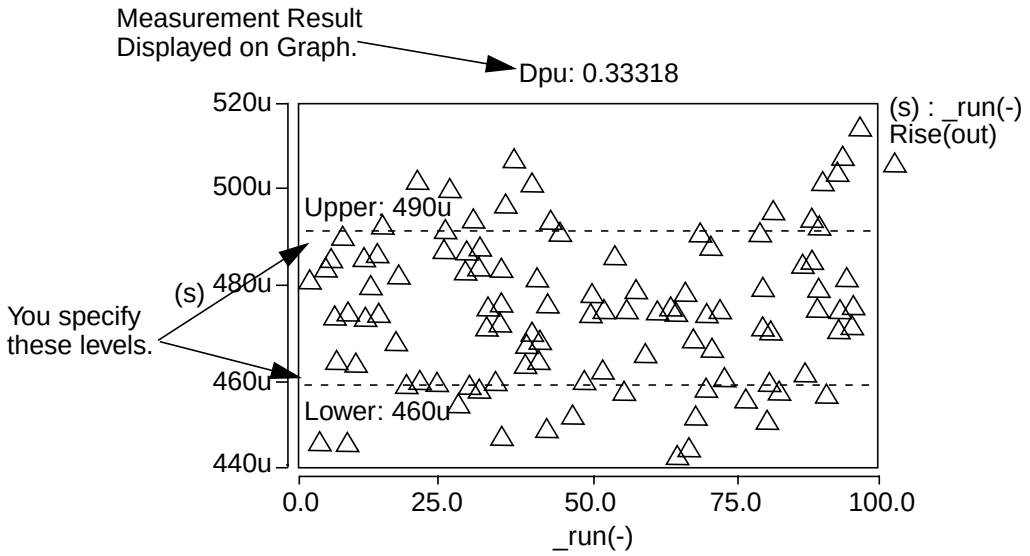
Type of Measured Waveform

Scatter plot

Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Dpu item and any other items you want to measure.
Specification Limits	Required values you provide.
Upper	Specifies upper measurement limit.
Lower	Specifies lower measurement limit.

Example



Duty Cycle

Description

Displays the duty cycle of a periodic waveform relative to default or specified topline and baseline levels.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog, digital

Possible Errors

An error is reported if the waveform does not contain at least one complete cycle.

Dialog Box Fields

Reference Levels

The following fields set the topline and baseline levels for the measured signal. You can display either of these levels on the waveform by clicking on the Visibility Indicator at the right of each field.

Topline
Specify a topline value within the upper and lower Y-axis values, or use the default value.

Baseline
Specify a baseline value below the topline value that is within the Y-axis values, or use the default value.

Trigger

⎍⎍/⎍⎍ Specifies that the measurement starts from a period with either a rising or falling edge.

⎍⎍ Specifies that the measurement starts from a period with a rising edge.

⎍⎍ Specifies that the measurement starts from a period with a falling edge.

Create New Waveform on Active Graph or New Graph

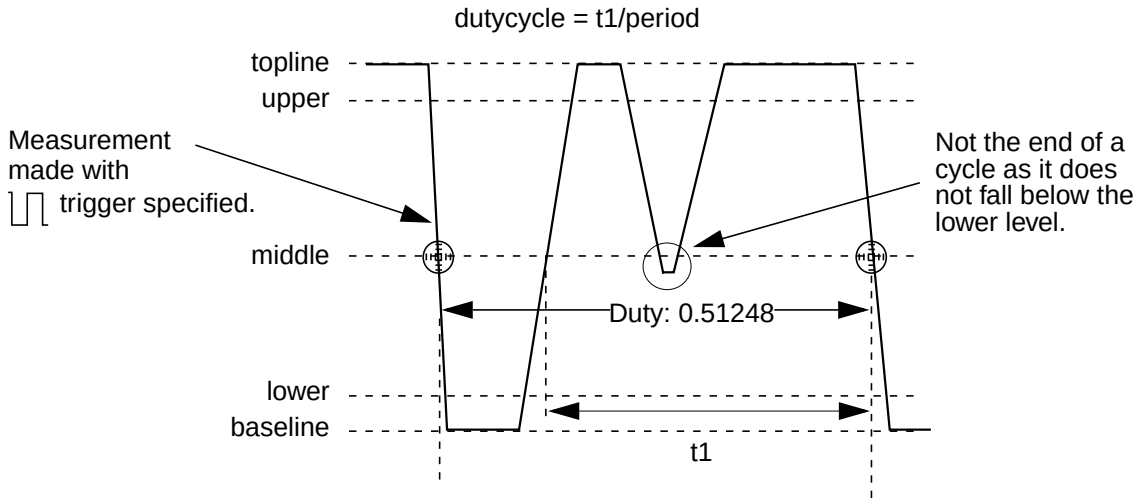
Duty Cycle vs. t - A new waveform is computed with duty cycle (Y-axis) versus time (X-axis).

Duty Cycle Calculation

The duty cycle is calculated as the ratio of the “high” portion of the waveform to the length of the period. In the example, the duty cycle is t1/period. (The circled portion of the waveform in the example is considered “high” and does not influence the calculation since it does not fall below the *lower* level of the waveform.)

Further details of how the period is measured can be found in the description for the Period measurement. More information about reference levels is provided in “Waveform Reference Levels”.

Example



Eye Diagram

Description

An eye diagram is used to display the behavior of a waveform cycle during a specific period of time. The eye diagram Measurement dialog has the ability to effectively overlap periods of time within a specified periodic waveform.

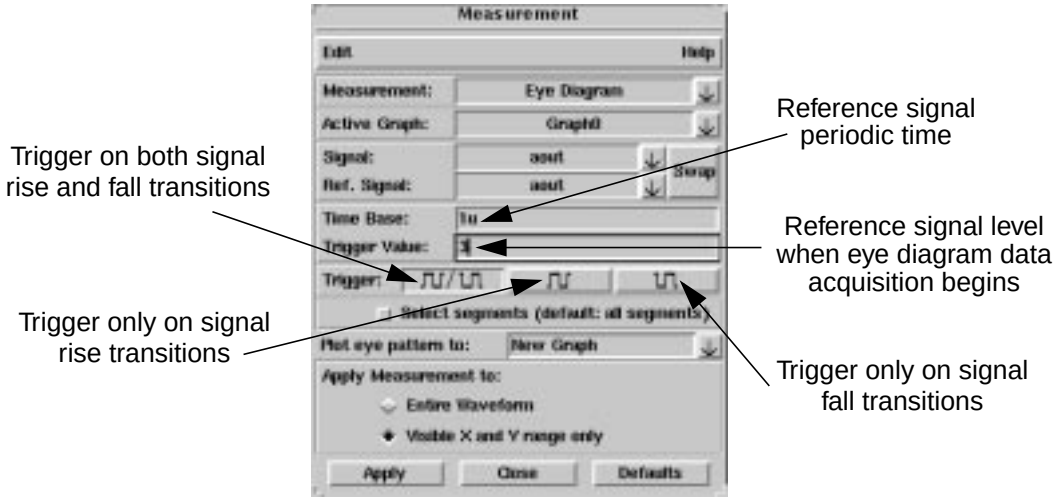
This type of periodic waveform display provides signal analysis characteristics much like the measurable variations found in periodicity or jitter output.

There are two options to apply Eye Diagram measurement:

- A reference signal can be used to define when data is acquired from the output signal and will be include in the eye diagram for calculation. This is the default way.
- A time base, start x value and end x value are specified by user. This is called Ideal trigger Eye Diagram.

Example (option 1)

The Eye Diagram Measurement dialog box, as shown in the following figure, can be accessed by choosing the
Tools > Measurement > Time Domain > Eye Diagram menu item.

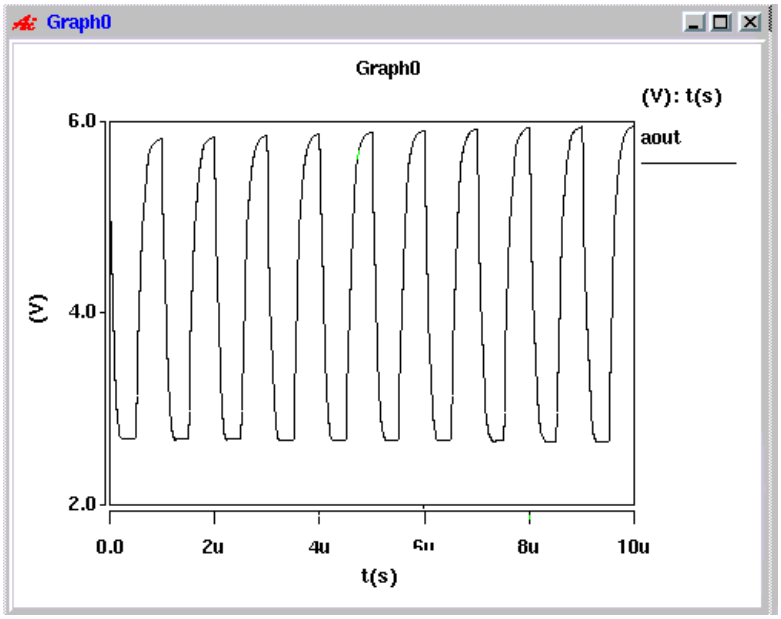


Field names in the Eye Diagram Measurement dialogue:

- **Time Base:** Indicates the time period elected for the reference signal.
- **Trigger Value:** Indicates at what level the reference signal is set when data begins to accumulate.
- **Trigger:** Specifies that the measurement trigger can start from either a rising state only, a fall state only, or both signal rise and fall state transitions.

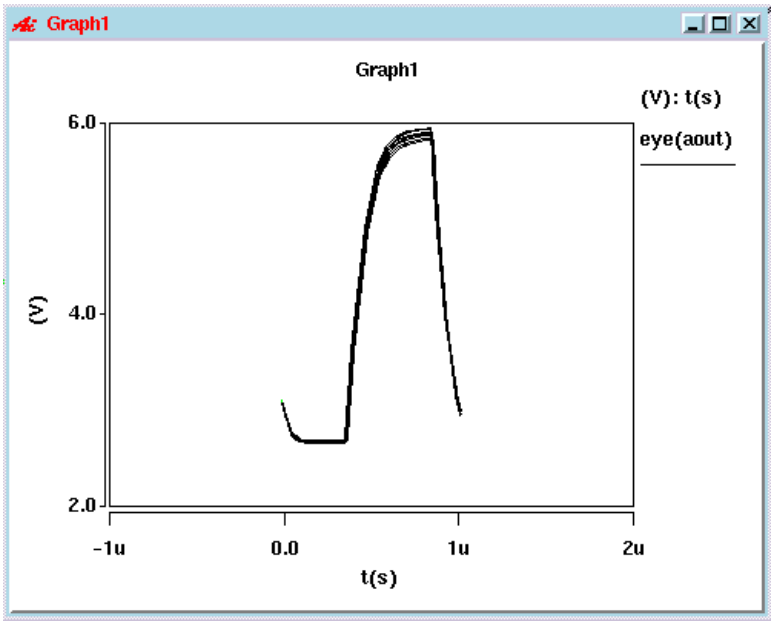
The eye diagram measurement has the ability to be applied to the entire waveform or exclusively to the visible X and Y range of the output signal.

The following figure shows the `aout` waveform that has been traced from the `single_amp.tr.ai_pl` plotfile. This example can be found in the `saber_amp` examples directory.

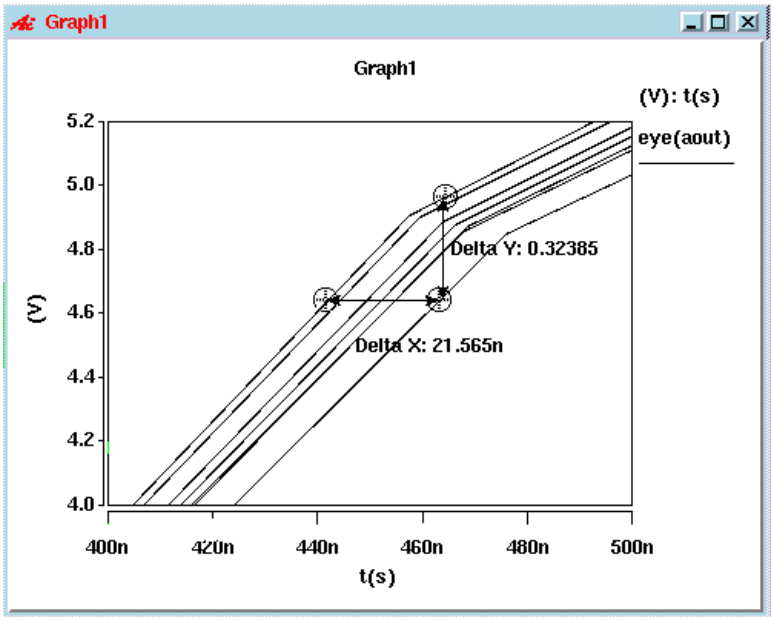


The Measurement form shown above indicates that the eye diagram has been setup on the `aout` waveform.

The following figure shows the output of the signal in the aout waveform after the eye diagram measurement has occurred.



The figure below shows the signal gain after applying delta X and Y measurements to a portion of the output signal.



Example (option 2)

The Eye Diagram Measurement dialog box can be accessed by choosing the **Tools > Measurement > Time Domain > Eye Diagram** menu item and selecting the **Ideal Trigger** radio button.

Field names in the Eye Diagram Measurement dialog:

- Time Base: Indicates the time period elected for the signal
- Start X value: Indicates the start time (optional)
- End X value: Indicates the end time (optional)

The eye diagram measurement can be applied to the entire waveform or exclusively to the visible X and Y range of the output signal.

Measurement Category

Time Domain

Type of Measured Waveform

Analog

Eye Mask

Description

Eye Mask is used to display the maximum width and height of an eye diagram opening.

Eye Mask measurement provides three Eye Mask types. They are Diamond Eye Mask, Rectangle Eye Mask, and Hexagon Eye Mask.

Measurement Category

Time domain

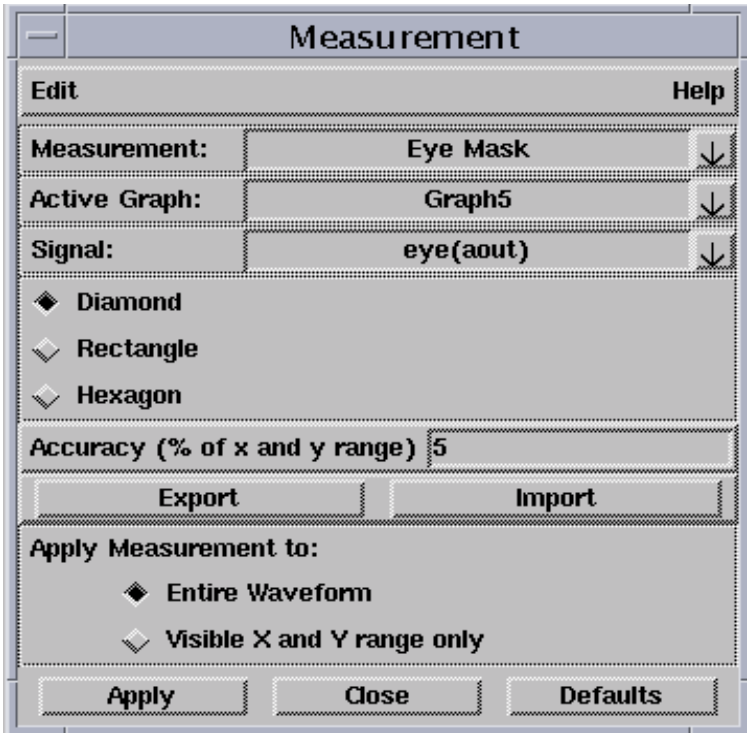
Type of Measured Waveform

Analog

Example

The Eye Mask Measurement dialog box, as shown in the following figure, can be accessed by choosing the menu item:

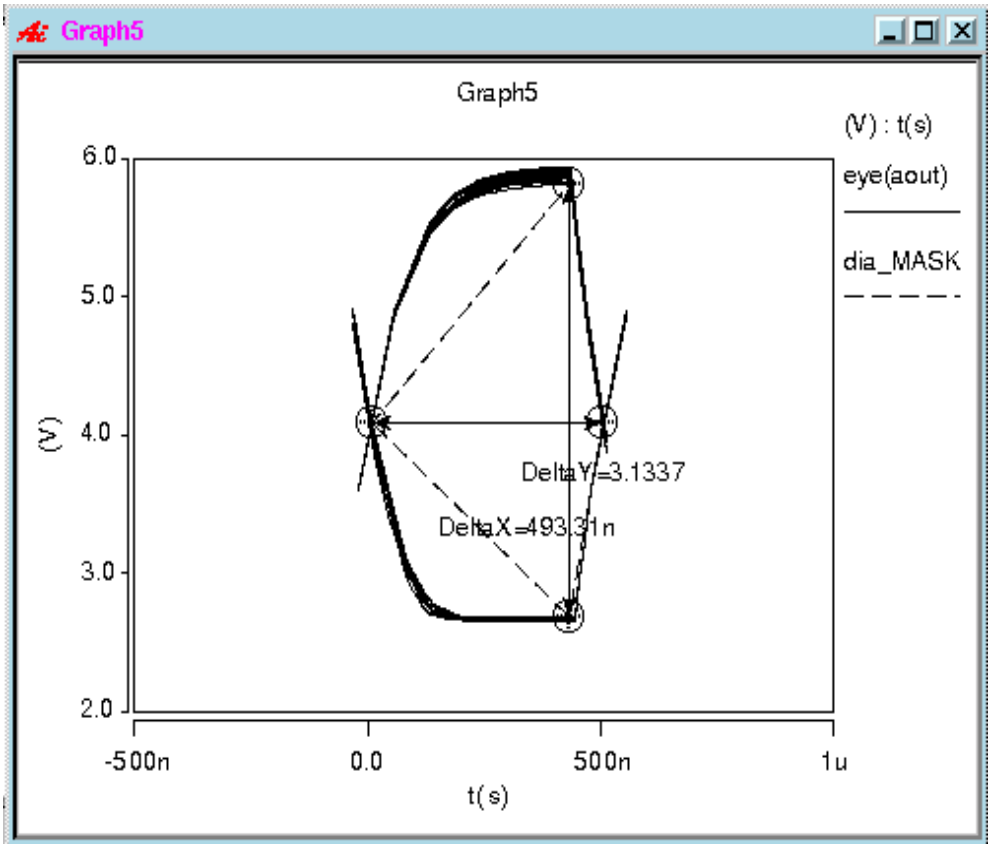
Tools > Measurement > Time Domain > Eye Mask



The field names in the Eye Mask Measurement dialogue are:

- Eye Mask type:
  In figure above, the Diamond radio button is selected.
- Accuracy (% of x and y range):
  Indicate the percentage of the x and y ranges within which, the Diamond Mask is calculated.
- When Hexagon or Rectangle radio button is selected:
  Hexagon's/Rectangle's extreme y-value: Indicate the maximum or minimum y value for the Hexagon/Rectangle Mask to be located at.
- Export:
  Export the masks on the current signal to a text file, *.mask.
- Import:
  Import a mask file and apply the masks in the file to the current signal.

The following figure is the Diamond Eye Mask with accuracy of x and y range 5% as set in the previous figure:



Falltime

Description

Displays the falltime between selected upper/lower levels of a waveform. You can also compute the falltime based on manually-set upper/lower levels as described in the topic titled Manually Set a Custom Topline/Baseline.

Measurement Category

Time Domain

Type of Measured Waveform

Analog

Possible Errors

An error is reported if the waveform contains no falling edges.

Dialog Box Fields

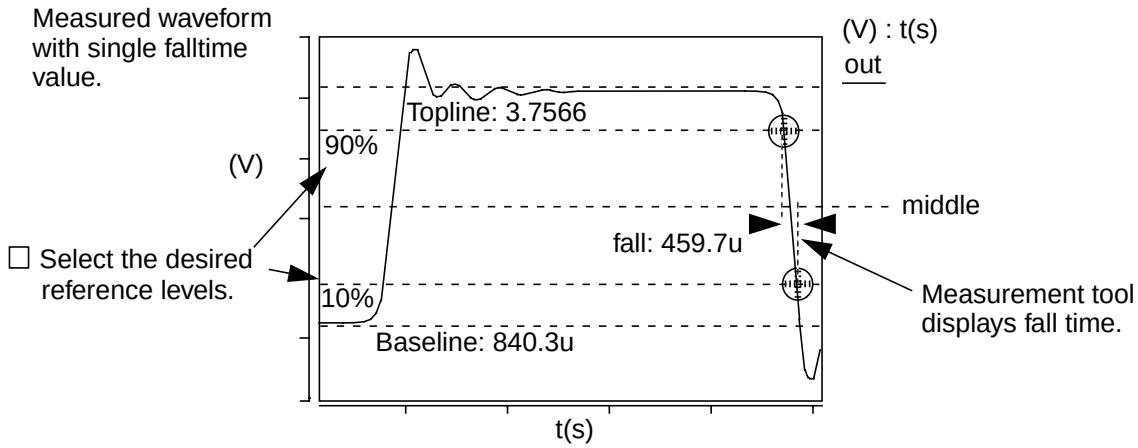
Reference Levels	The following fields set the topline and baseline levels for the measured signal. You can display either of these levels on the waveform by clicking on the Visibility Indicator at the right of each field. Topline Specify a topline value or use the default value. Baseline Specify a baseline value below the topline value or use the default value.
0-100% 10-90% 20-80%	Click on one of these buttons to set an *upper* and *lower* range (in percent) relative to the topline/baseline levels. To compute a falltime based on a different percentage level than the defaults, refer to the topic titled Manually Set a Custom Topline/Baseline.
Create New Waveform on Active Graph or New Graph	Falltime vs. t - Creates a new waveform of falltime value(s) (Y-axis) versus elapsed time (X-axis). See Example 2.

Falltime Calculation

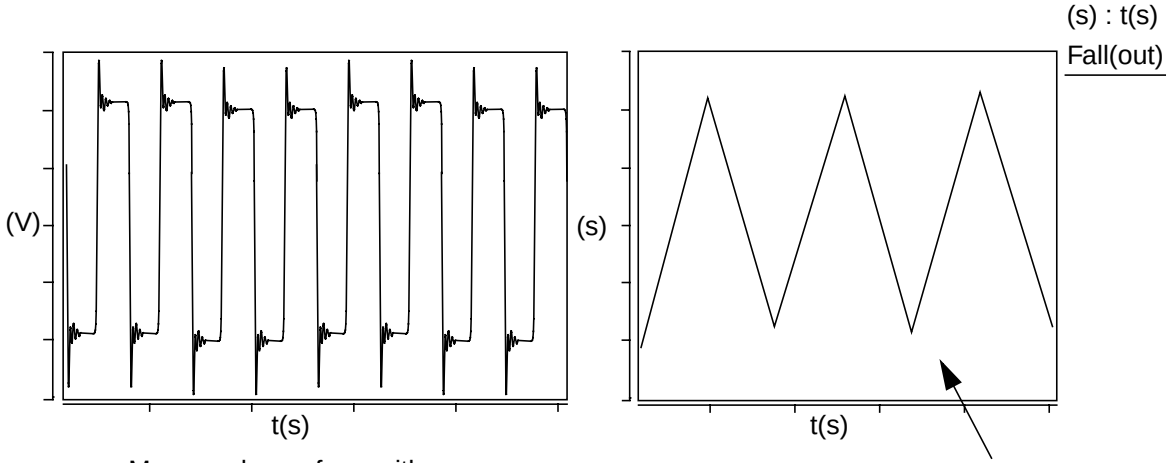
The falltime is calculated by finding a crossing with the *middle* level of the waveform. Looking forward from this point, the time when the waveform falls to the *lower* level is found. Looking backward, the time when the waveform rises to the *upper* level is also found. You set the *upper* and *lower* levels as 0-100%, 10-90%, or 20-80% of the Topline/Baseline levels. The difference in the times is the falltime, as shown in the example.

For more information about the *lower, middle,* and *reference* levels of a waveform, refer to "Waveform Reference Levels".

Example 1



Example 2



Frequency

Description

Displays the frequency of a periodic waveform relative to default or specified topline and baseline levels.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog, digital

Possible Errors

An error is reported if the waveform does not contain at least one complete cycle.

Dialog Box Fields

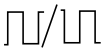
Reference Levels

The following fields set the topline and baseline levels for the measured signal. You can display either of these levels on the waveform by clicking on the Visibility Indicator at the right of each field.

Topline
Specify a topline value or use the default value.

Baseline
Specify a baseline value below the topline value or use the default value.

Trigger


Specifies that the measurement starts from a period with either a rising or falling edge.

Specifies that the measurement starts from a period with a rising edge.

Specifies that the measurement starts from a period with a falling edge.

Create New Waveform on Active Graph or New Graph

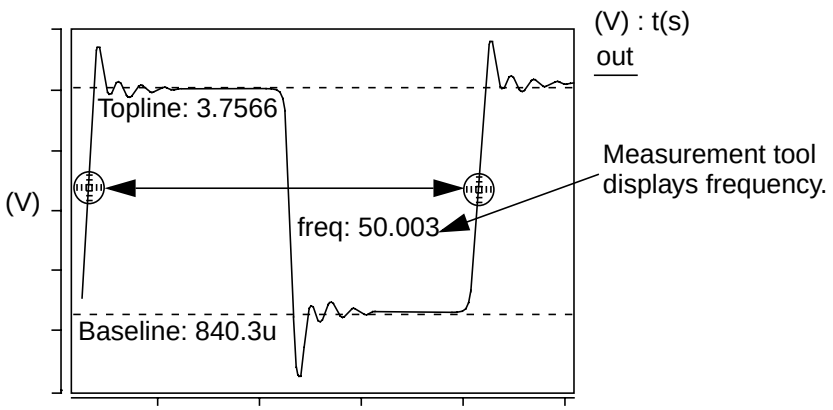
Frequency vs. t - A new waveform is computed with frequency (Y-axis) versus time (X-axis). See Example 2.

Frequency Calculation

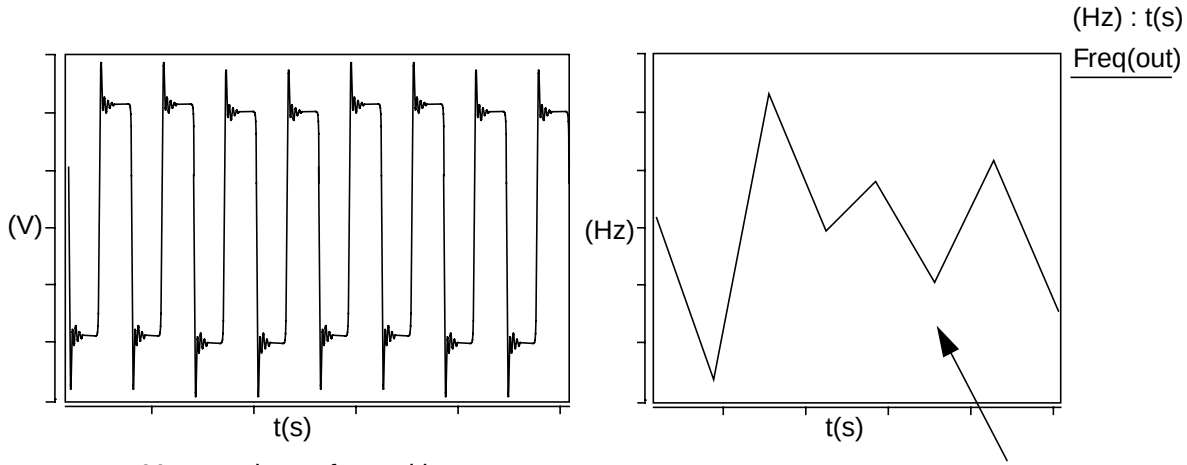
The frequency is calculated as the reciprocal of the period. For information on how the period is computed, refer to the Period Calculation.

Example 1

Measured waveform with single frequency value.



Example 2



Measured waveform with multiple frequency values.

Measurement tool produces this waveform showing frequency as a function of time.

Gain Margin

Description

Displays the gain margin in dB of a complex waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

Possible Errors

An error is reported if the phase of the measured waveform does not pass through –180 degrees or if the waveform is not complex.

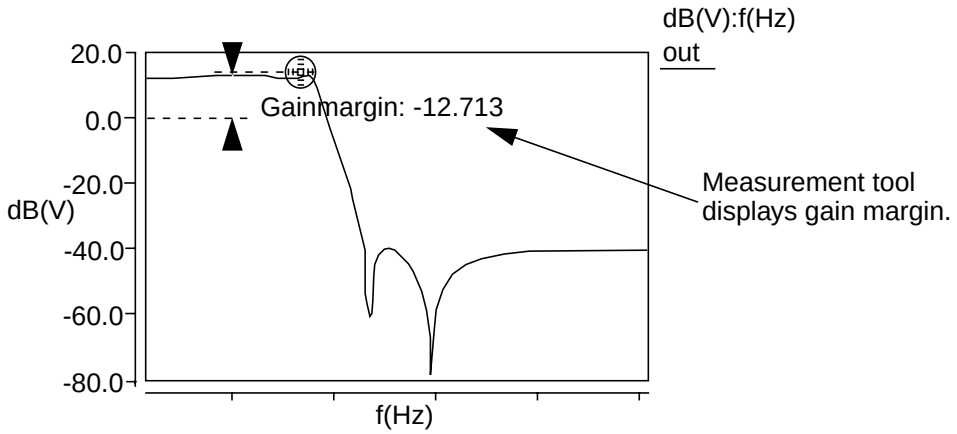
Dialog Box Fields

There are no unique fields in this form.

Gain Margin Calculation

The gain margin is defined as the difference between the gain of the measured waveform and 0 dB at the frequency where the phase shift is –180 degrees.

Example



Highpass

Description

Displays the corner frequency of a waveform with a highpass shape. The measurement is made relative to a default or specified topline level and a specified offset.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog

Dialog Box Fields

Reference Levels — If you want to see the topline or offset level, or both, displayed on the waveform, click on the Visibility Indicator to the right the Topline or Offset field.

Topline
You set this field to a default or a specified level.

Offset
You specify an offset value, to be applied relative to the Topline value. The default is 3. You must also choose which operator to use (–, +, *, or /) along with the specified level. The default is the minus sign. This resulting level is also called the measurement level.

Highpass Calculation

The corner frequency is found by searching from right to left until the waveform first falls below the measurement level, which is determined by the offset (from the topline) that you specify.

Topline

If you do not specify the topline, a default value is calculated by using a method specified in the Default Topline/Baseline field in the Measurement Preference dialog box.

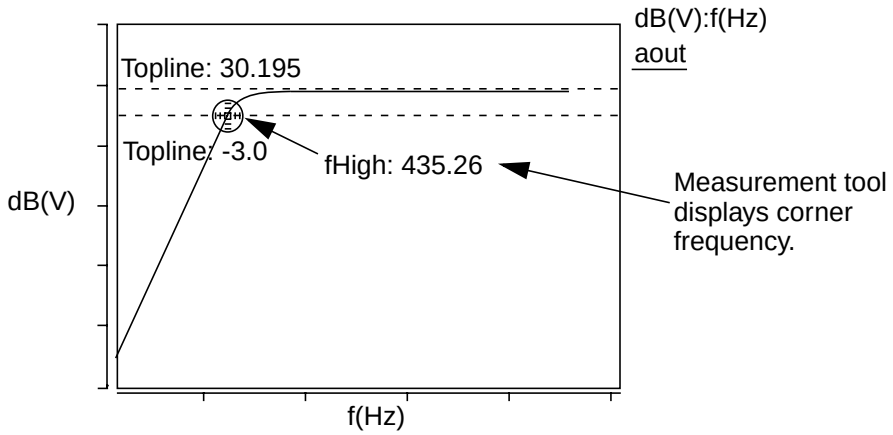
Offset

Computed as one of

```
topline - offset_value,
topline + offset_value,
topline * offset_value, or
topline / offset_value,
```

depending on which operator you choose. This level is also referred to as the Measurement Level, as shown in the Example.

Example



Histogram

Description

Displays a histogram of a waveform.

Histograms can display absolute values (such as the number of runs that fall into a certain range), where only integer numbers make sense. Normalized values can also be displayed (number of runs in a range divided by the total number of runs), where the values are fractions between 0 and 1. By design, the Y axis of histograms can have non-integer values.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, analog, event-driven analog

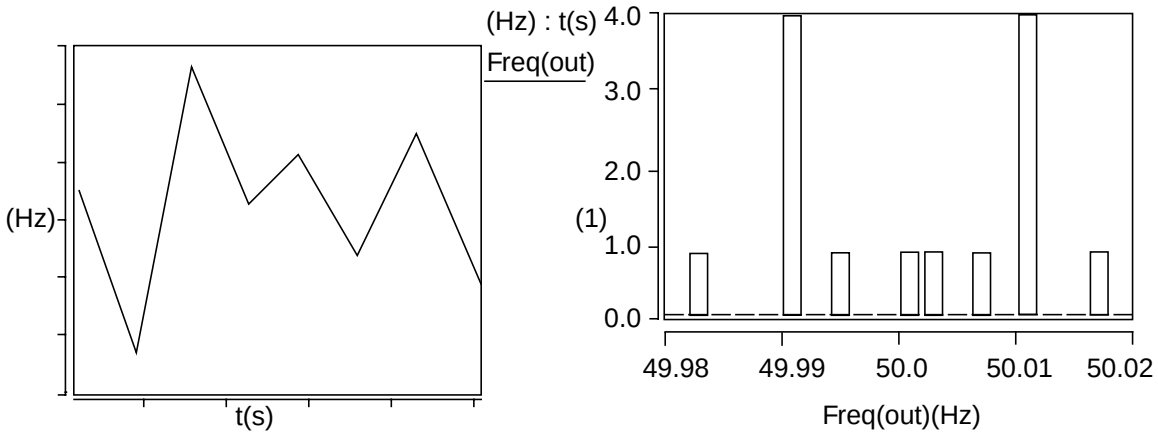
Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Histogram item and any other items you want to measure.

Example

Previous measurement result of multiple frequency values.

Histogram of previous measurement result.



Horizontal Level

Description

Displays a moveable horizontal line to identify Y-axis levels.

Measurement Category

General

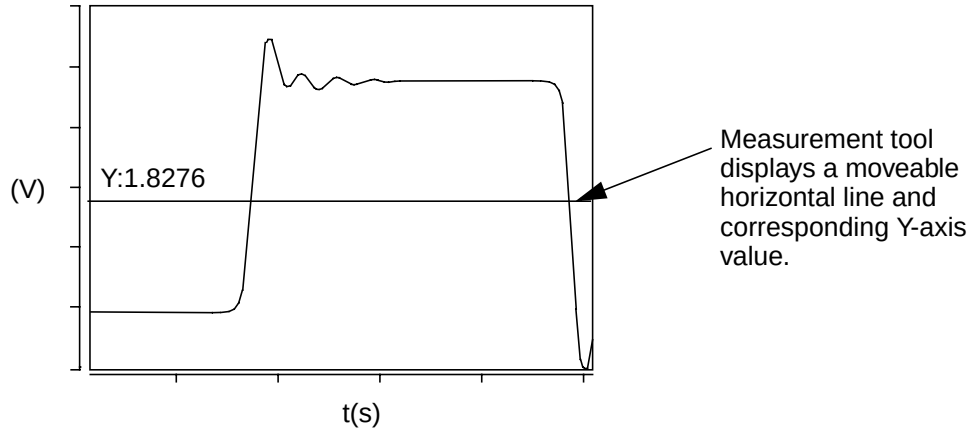
Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram, spectral

Dialog Box Fields

Location (Optional) Y Value	You can specify a Y-value to place a moveable horizontal line on the waveform. If you do not specify one, a default value is computed.

Example



Imaginary

Description

Displays the imaginary value of a point on a waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

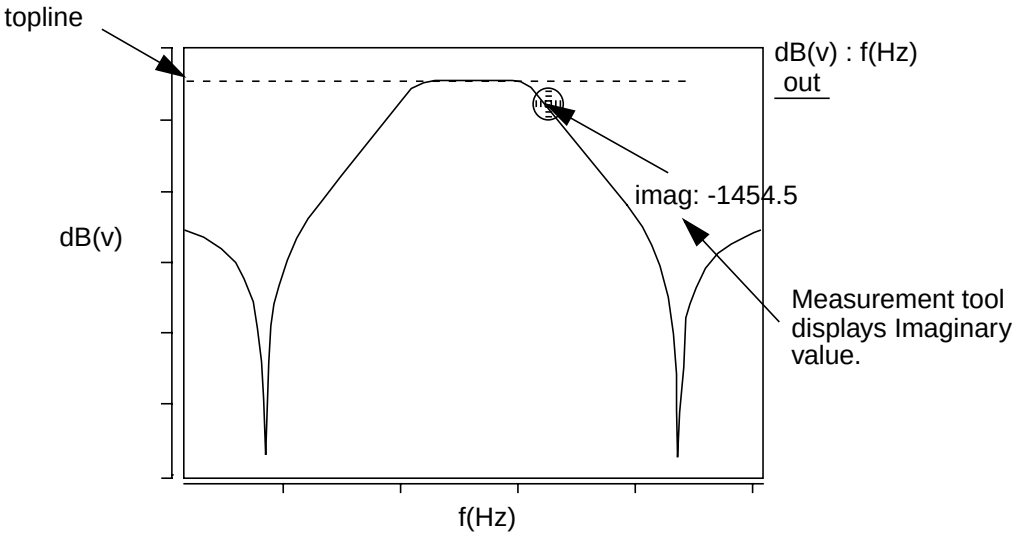
Dialog Box Fields

X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

Imaginary Calculation

The imaginary value of a waveform is the imaginary part of a complex argument. If there is no complex part then the value 0.0 is returned.

Example



IP2

Description

For a nonlinear system with an ouput and two fundamental excitation frequencies f1 and f2, the second order Intercept Point is the point on a set of PowerOut (PowerOut1 and PowerOut2) versus PowerIn curves at which a line extrapolated from PowerOut1 with a slope of 1, and a line with a slope of 2 extrapolated from PowerOut2 intersect. PowerOut1 represents the first order term, and PowerOut2 is the second order term. Both lines must be extrapolated from a region with sufficiently low input power. OIP2 is the value of PowerOut at which IP2 occurs, IIP2 is the value of the PowerIn value at which IP2 occurs.

Measurement Category

RF

Type of Measured Waveform

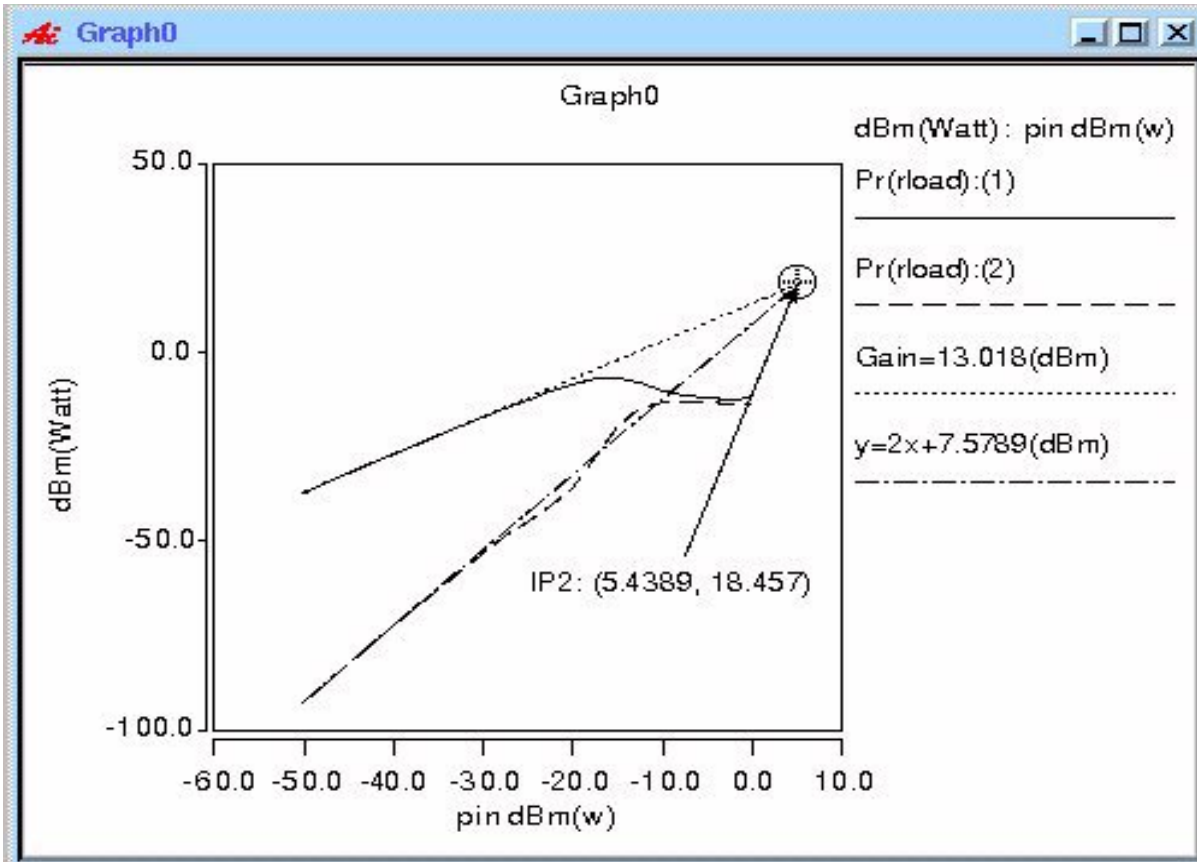
Analog, Power type, either PowerOut versus PowerIn or PowerOut versus Frequence with PowerIn as a sweep parameter.

Dalog Box Fields

Given two waveforms, one is PowerOut1 versus PowerIn, the other is PowerOut2 versus PowerIn, the following Dialog Box Fields show:

PowerOut1	First-order term PowerOut1 versus PowerIn curve
PowerOut2	Second-order term PowerOut2 versus PowerIn curve
PowerIn	PowerIn value

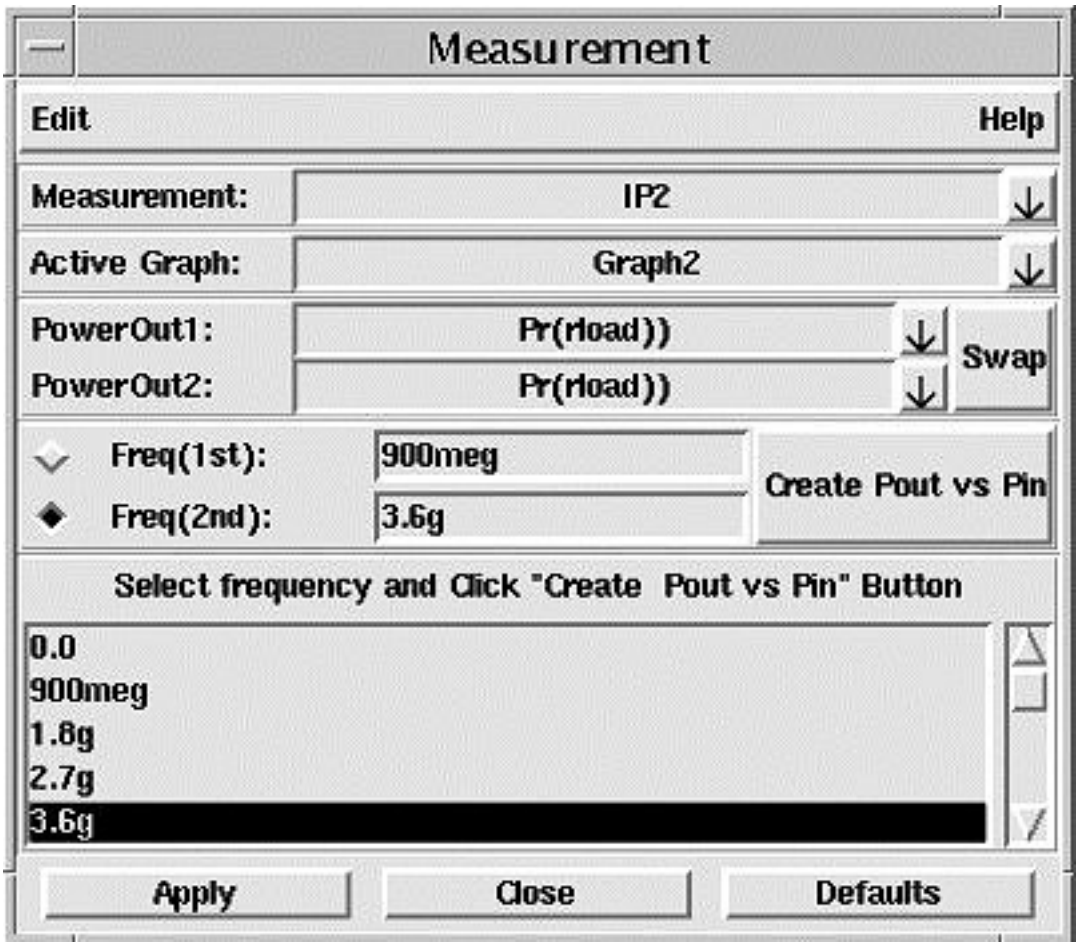
Examples

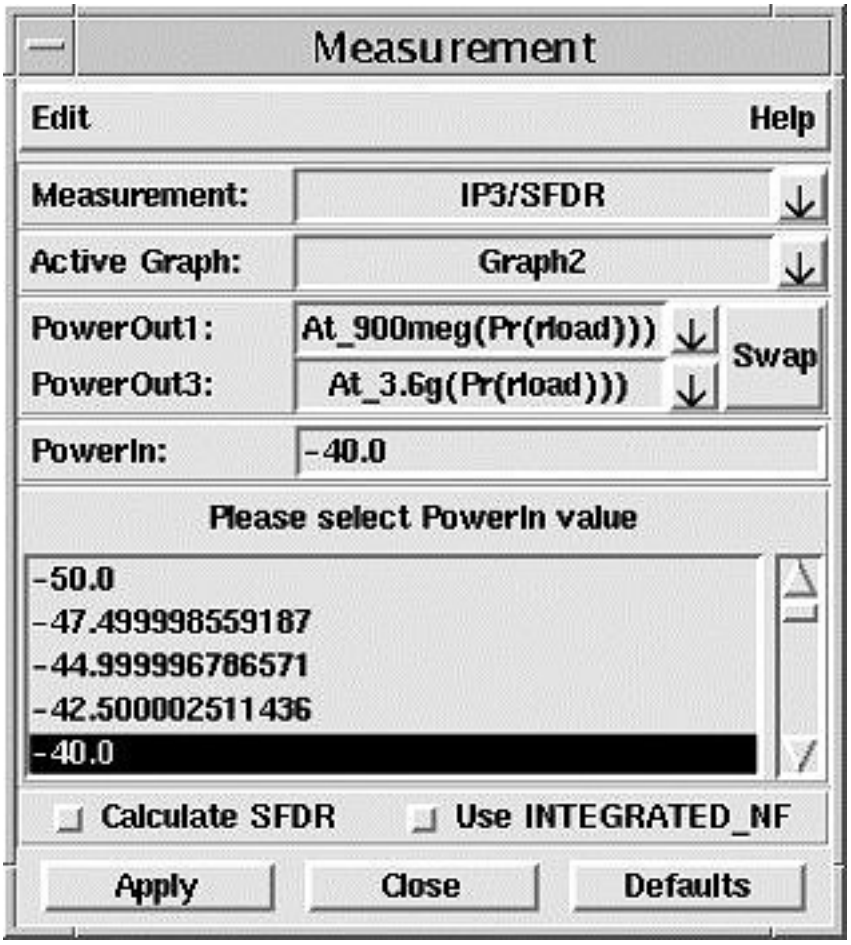


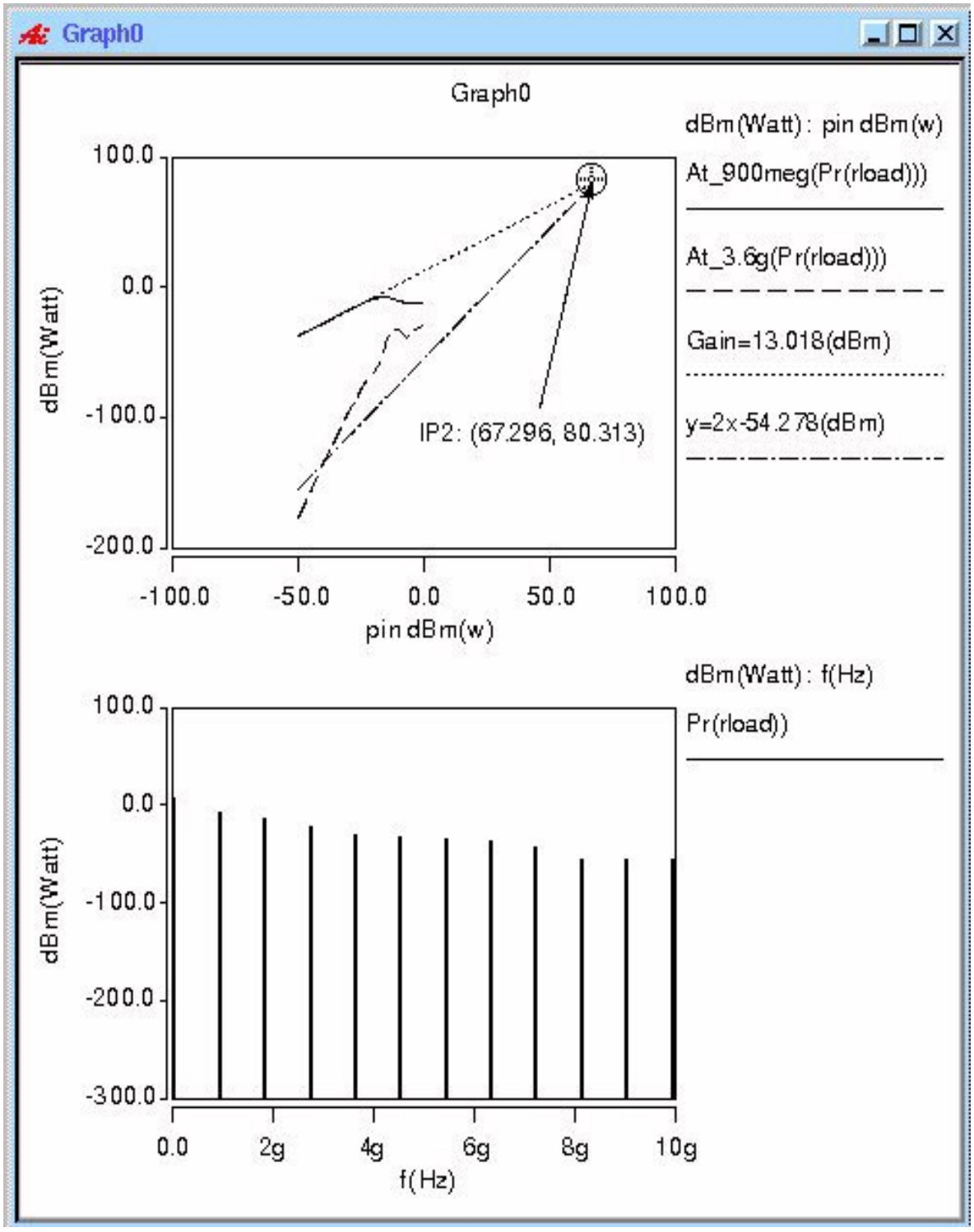
Given one waveform, PowerOut versus Frequency with PowerIn as a sweep parameter, the following Dialog Box Fields show:

PowerOut1	Waveform of PowerOut versus Frequency with PowerIn as a sweep parameter
PowerOut2	Second-order term PowerOut2 versus PowerIn curve
Freq (First)	Frequency to generate first-order term PowerOut1 versus PowerIn curve
Freq (Second)	Frequency to generate second-order term PowerOut2 versus PowerIn curve

Example







IP3/SFDR

Description

For a nonlinear system with an ouput and two fundamental excitation frequencies f1 and f2, the third order Intercept Point is the point on a set of PowerOut (PowerOut1 and PowerOut3) versus PowerIn curves at which a line extrapolated from PowerOut1 with a slope of 1, and a line with a slope of 3 extrapolated from PowerOut3 intersect. PowerOut1 represents the first order term, and PowerOut3 is the third order term. Both lines must be extrapolated from a region with sufficiently low input power. OIP3 is the value of PowerOut at which IP3 occurs, IIP3 is the value of the PowerIn value at which IP3 occurs.

Measurement Category

RF

Type of Measured Waveform

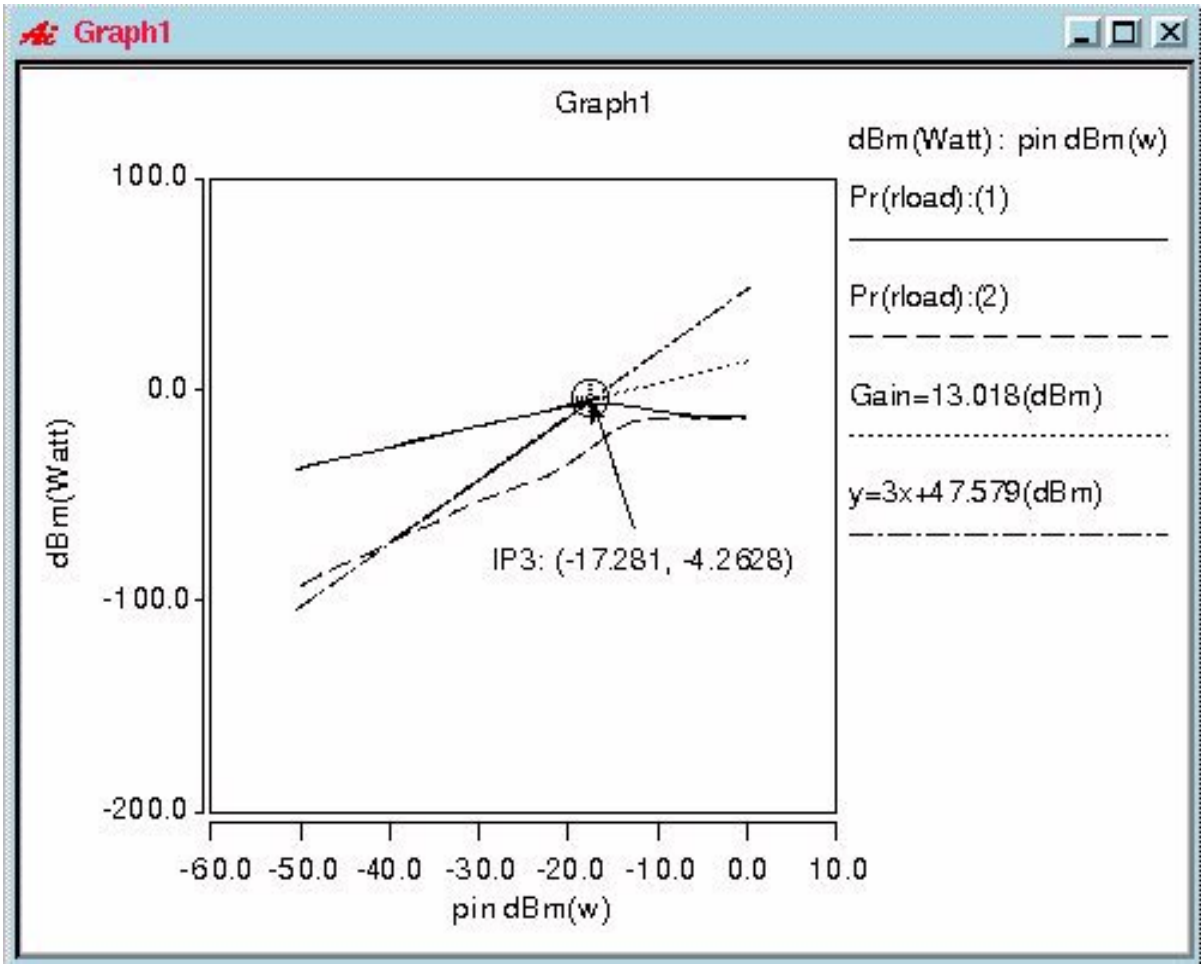
Analog, Power type, either PowerOut versus PowerIn or PowerOut versus Frequence with PowerIn as a sweep parameter.

Dalog Box Fields

Given two waveforms, one is PowerOut1 versus PowerIn, the other is PowerOut3 versus PowerIn, the following Dialog Box Fields show:

PowerOut1	First-order term PowerOut1 versus PowerIn curve
PowerOut3	Third-order term PowerOut3 versus PowerIn curve
PowerIn	PowerIn value

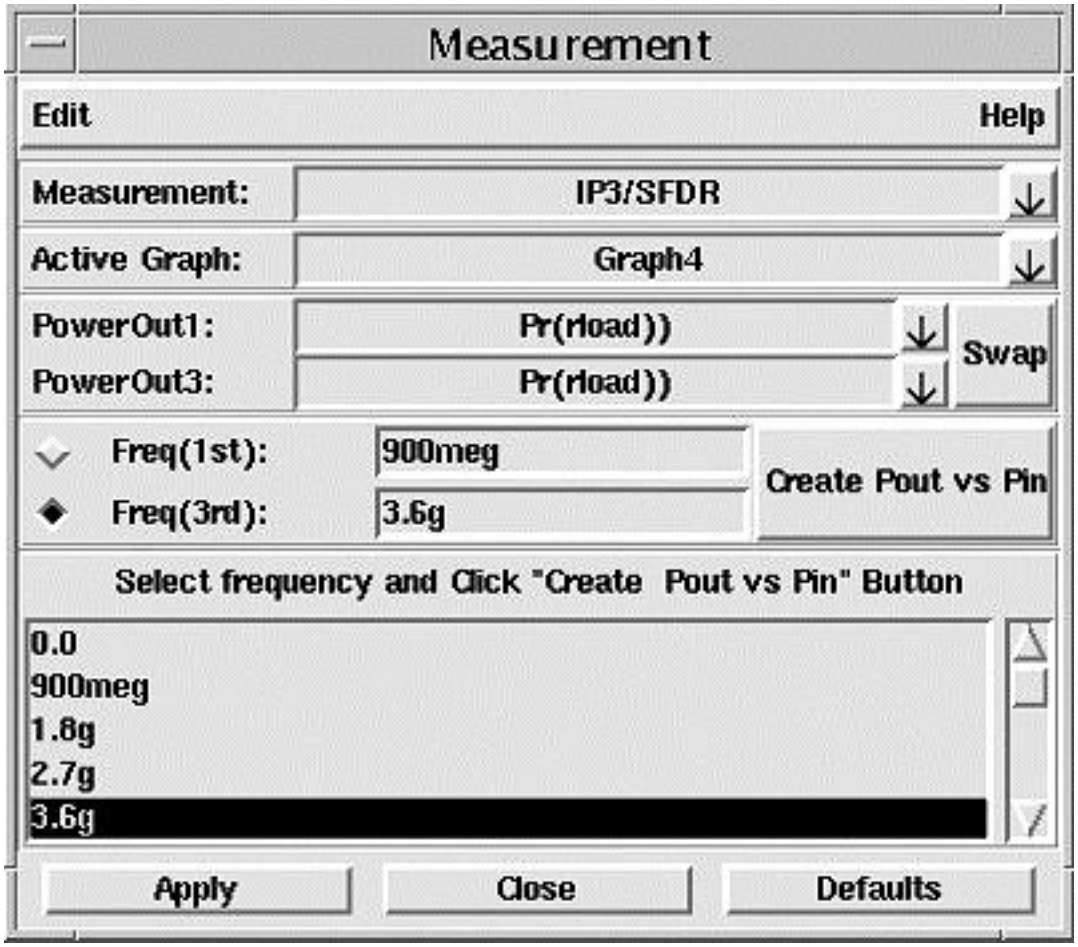
Examples

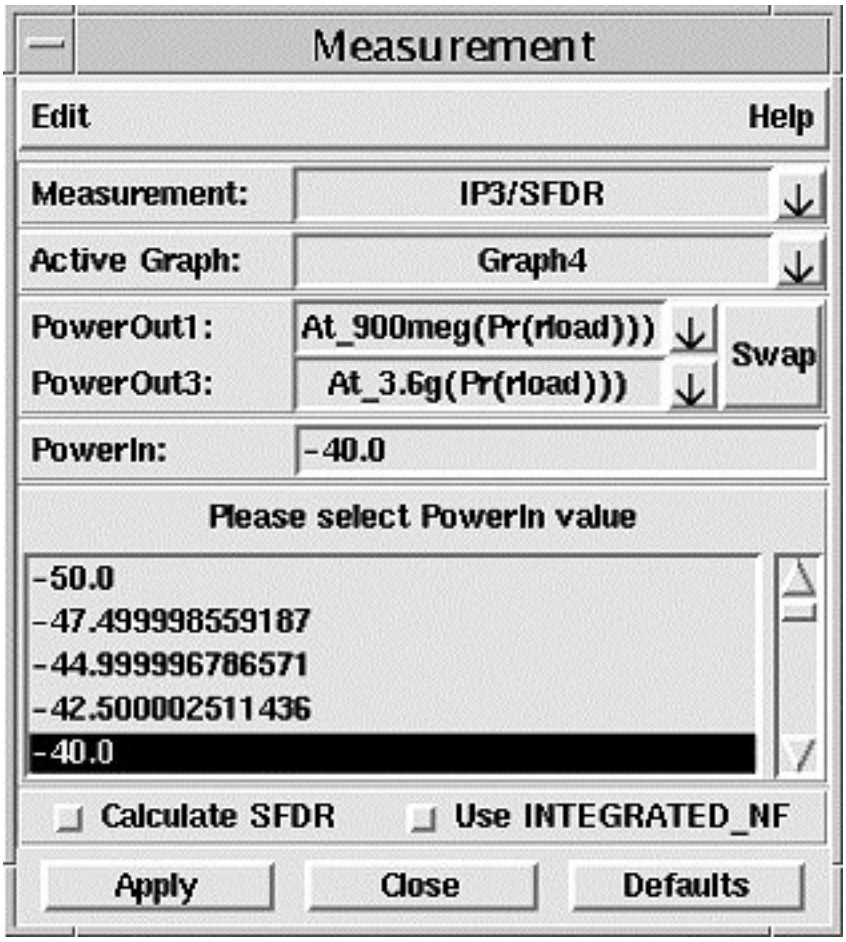


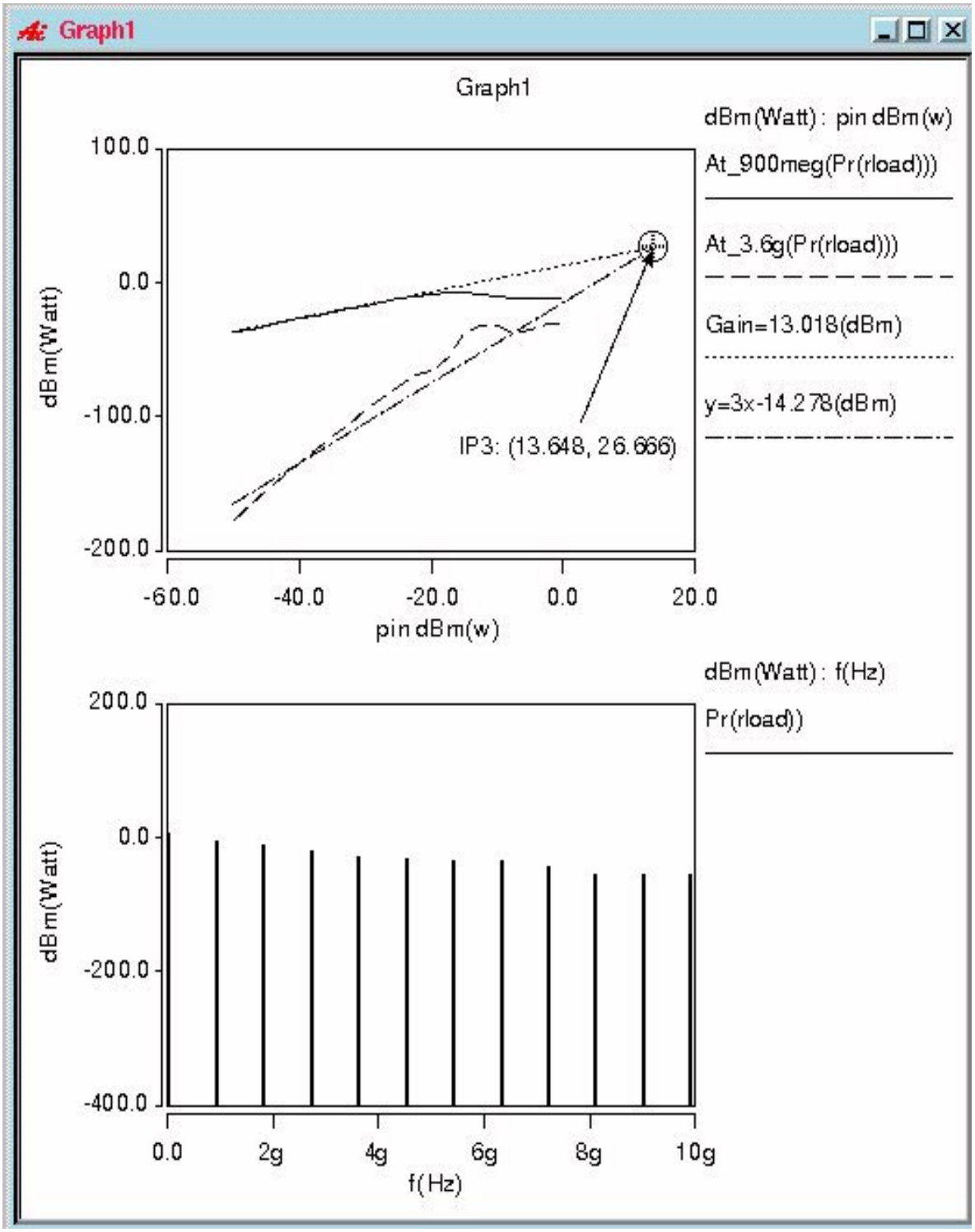
Given one waveform, PowerOut versus Frequency with PowerIn as a sweep parameter, the following Dialog Box Fields show:

PowerOut1	Waveform of PowerOut versus Frequency with PowerIn as a sweep parameter
PowerOut3	This field is not used in this case
Freq (First)	Frequency to generate first-order term PowerOut1 versus PowerIn curve
Freq (Second)	Frequency to generate third-order term PowerOut3 versus PowerIn curve

Example







Length

Description

Displays the length of a straight line that connects two X-axis points on a waveform or two X-axis points on two waveforms. If two waveforms are selected, the two waveforms do not need to be the same type, but they must be in the same graph region.

Measurement Category

General

Type of Measured Waveform

Analog, event-driven analog, scatter plot

Dialog Box Fields

Anchor Snap Signal
: Generates moveable anchor points that snap to the signal waveform. See the Example.

  Closest Measurement
  : When you have multiple anchor points visible on a graph from previous measurements, this setting causes the measurement to snap to one of those nearest points. If there is no visible measurement on the active graph from which to snap, an error message is displayed. To see an example, refer to the Point Marker Example.

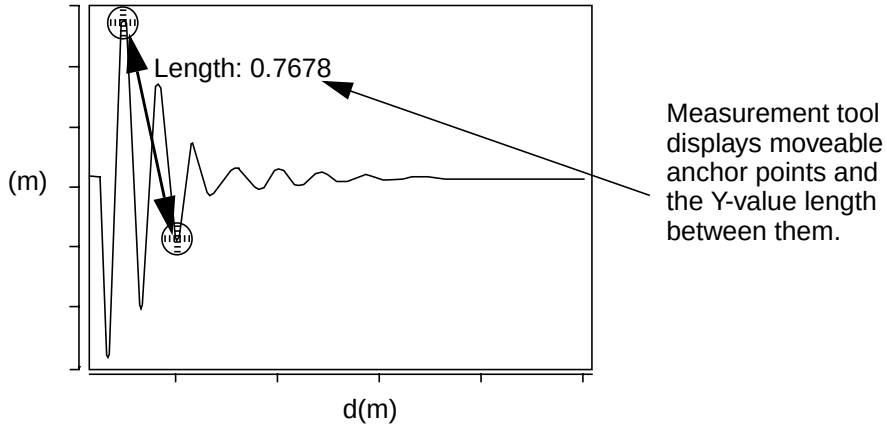
  Floating
  : Generates moveable anchor points are float within the graph. To see an example, refer to the Delta Y Example.

Location (Optional)
: You can optionally specify two X-values that are used to determine the Y-value difference between the two X-values.

X1 Value
: Optional value. You can change it by moving the anchor point after the initial measurement is made.

X2 Value
: Optional value. You can change it by moving the anchor point after the initial measurement is made.

Example



Local Max/Min

Description

Displays the local maximum or minimum point, or both, on a waveform.

Measurement Category

General

Type of Measured Waveform

Analog, event-driven analog

Possible Errors

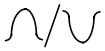
An error is reported if a local minimum or maximum is not found within the waveform.

Dialog Box Fields

Peak Type


All local maximum and local minimum points are found.


Only the local maximum points are found.


Only the local minimum points are found.

Peak Threshold

{value} % of peak to peak -
You specify a value in percent of the peak to peak. Peaks below this level are rejected by the measurement. The default setting is 2%.

Create New Waveform on Active Graph or New Graph

The peak type you have selected will determine which fields are displayed under this heading as follows:


Local Extreme vs. t -
A new waveform is plotted by connecting the Y-values found at the extreme points (Y-axis) against time (X-axis).

X at Local Extreme vs. t -
A new waveform is plotted by connecting the X-axis points found (Y-axis) against time (X-axis).

Local Maximum vs. t -
A new waveform is plotted by connecting the Y-values found at the maximum points (Y-axis) against time (X-axis).

X at Local Maximum vs. t -
A new waveform is plotted by connecting the X-axis points found (Y-axis) against time (X-axis).

Local Minimum vs. t -
A new waveform is plotted by connecting the Y-values found at the minimum points (Y-axis) against time (X-axis).

X at Local Minimum vs. t -
A new waveform is plotted by connecting the X-axis points found (Y-axis) against time (X-axis).

Local Min/Max Calculation

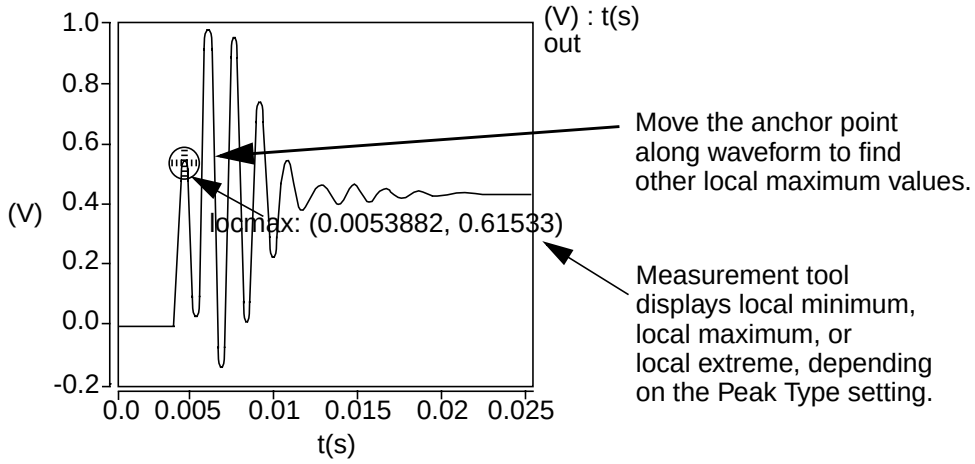
 A local maximum occurs when the slope of the waveform changes sign from positive to negative.

 A local minimum occurs when the slope of the waveform changes sign from negative to positive.

The end points of the waveform are not considered for either calculation.

The Peak Threshold value you specify determines whether a point at which a positive-to-negative or negative-to-positive slope change occurs should be selected as a local maximum or local minimum. The Peak Threshold value specifies the minimum change between an adjacent local maximum and local minimum relative to the peak-to-peak value of the waveform.

Example



Multi-Member Waveform Options

Create New Waveform on Active Graph or New Graph

Creates a new waveform in the **Active Graph** or in a **New Graph** to display the results.

Local Extreme or Maximum or Minimum vs. _run or *Vary_Parameter* -
A scatter plot or analog waveform is displayed, which shows all the points found (Y-axis) against either the Monte-Carlo run (X-axis) or the Vary parameter value (X-axis).

X at Local Extreme or Maximum or Minimum vs. _run or *Vary_Parameter* -
A scatter plot or analog waveform is displayed, which shows all the points found (Y-axis) against either the Monte-Carlo run (X-axis) or the Vary parameter value (X-axis).

Local Extreme or Maximum or Minimum Histogram -
A histogram is plotted, which shows how many points were found (count, Y-axis) at each Y-value (X-axis).

X at Local Extreme or Maximum or Minimum Histogram -
A histogram is plotted, which shows how many points were found (count, Y-axis) at each X-value (X-axis).

Lowpass

Description

Displays the corner frequency of a waveform with a lowpass shape. The measurement is made relative to a default or specified topline and a specified offset.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog

Dialog Box Fields

Reference Levels

If you want to see the topline and/or offset level displayed on the waveform, click on the Visibility Indicator to the right the Topline or Offset field.

Topline
You set this field to a default or a specified level.

Offset
You specify an offset value, to be applied relative to the Topline value. The default is 3. You must also choose which operator to use (–, +, *, or /) along with the specified level. The default is the minus sign. This resulting level is also called the measurement level.

Lowpass Calculation

The corner frequency is found by searching from left to right until the waveform first falls below the measurement level, which is determined by the offset (from the topline) that you specify.

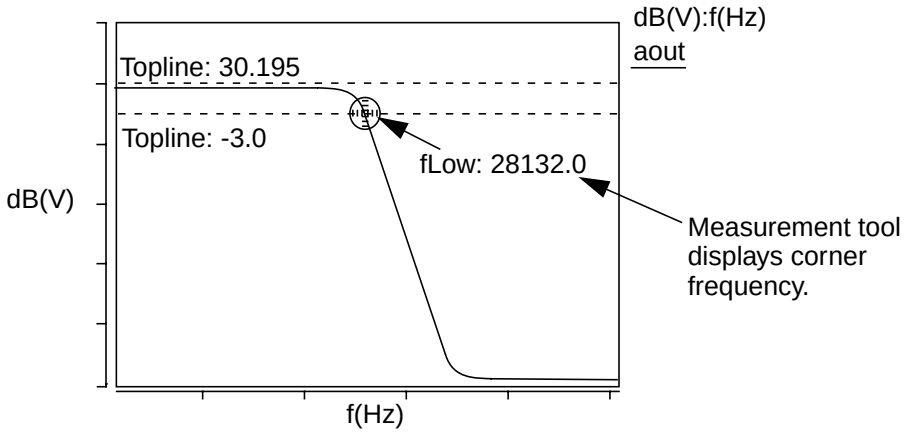
Topline

If you do not specify the topline, a default value is calculated by using a method specified in the Default Topline/Baseline field in the Measurement Preference dialog box.

Offset

Computed as one of
`topline - offset_value`,
`topline + offset_value`,
`topline * offset_value`, or
`topline / offset_value`,
depending on which operator you choose. This level is also referred to as the Measurement Level, as shown in the Example.

Example



Magnitude

Description

Displays the magnitude of a point on a waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

Dialog Box Fields

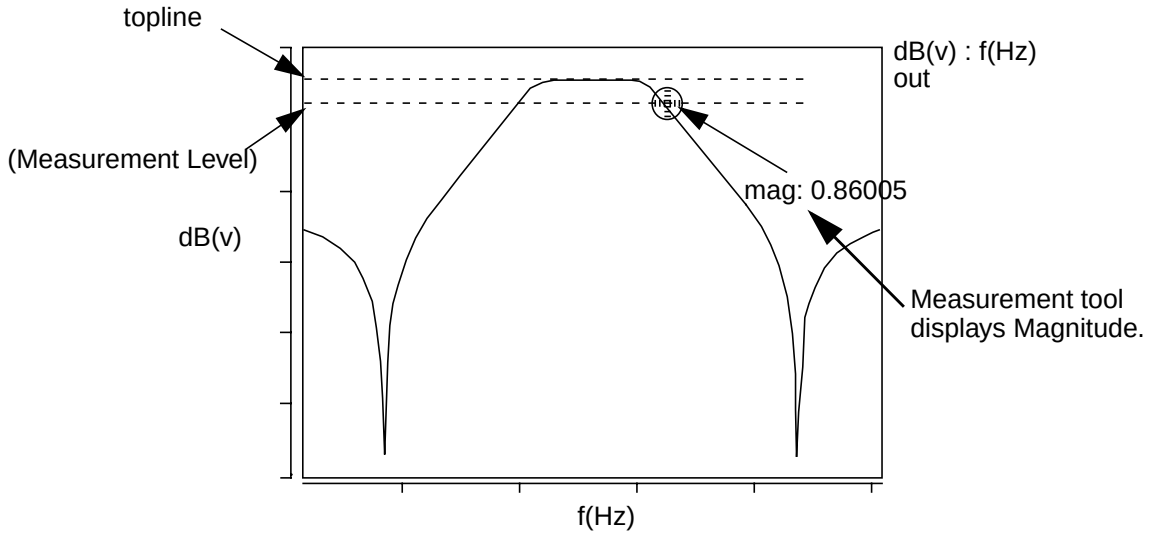
X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

Magnitude Calculation

The magnitude of a waveform is calculated as the absolute value of an argument

$$mag = \sqrt{(real^2 + imag^2)}$$

Example



Maximum

Description

Displays the maximum value of a waveform.

Measurement Category

Levels, Statistics

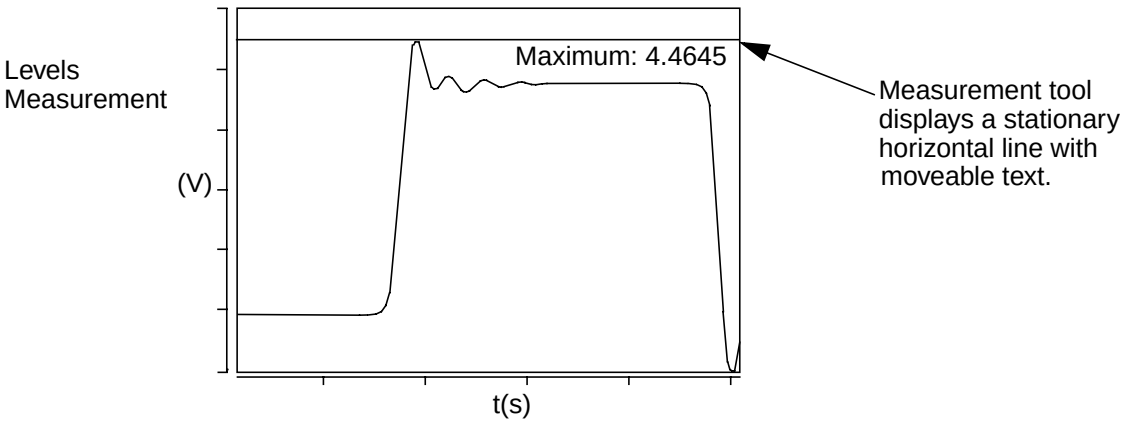
Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram, spectral

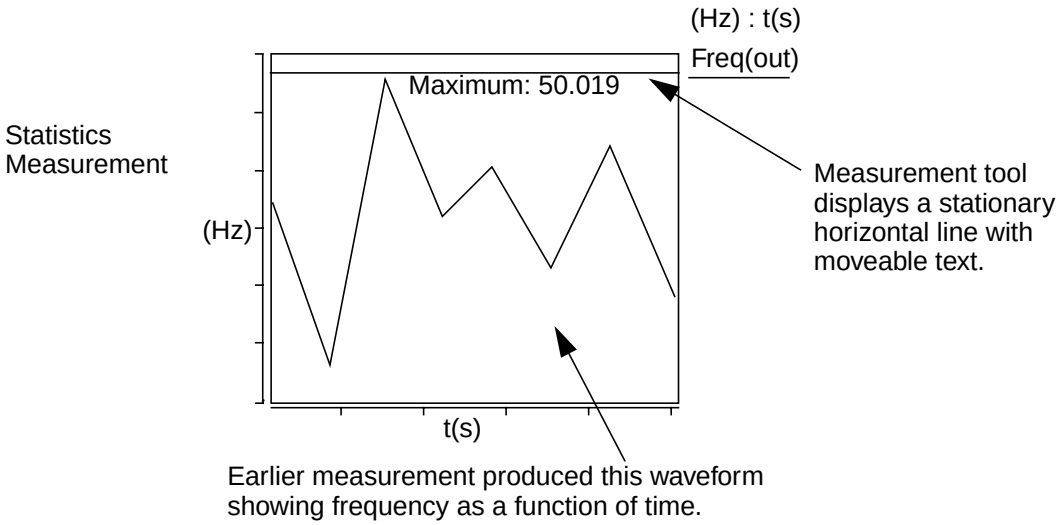
Dialog Box Fields

Category List — This measurement appears in the Statistics or Levels category. All Statistic or Levels category items appear below the Signal field (depending on which category you selected). Select the Maximum item and any other items you want to measure.

Example 1



Example 2



Mean

Description

Displays the mean value of a waveform.

Measurement Category

Statistics

Type of Measured Waveform

*S*catter plot, histogram, analog, event-driven analog

Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Mean and other items you want to measure.

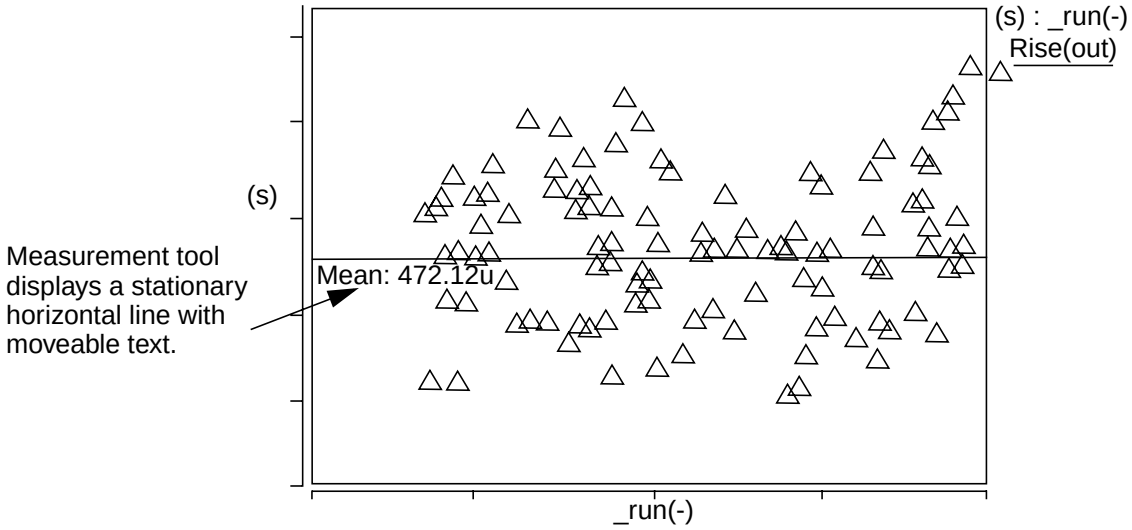
Mean Calculation

The mean value of a waveform is calculated as follows:

$$\frac{1}{N} \sum_{j=1}^{N} W_j$$

In this calculation, N is the number of points, and array W_j contains the individual points of the waveform.

Example



Mean +3 std_dev

Description

Displays the (mean + 3σ) value of a waveform.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, histogram, analog, event-driven analog

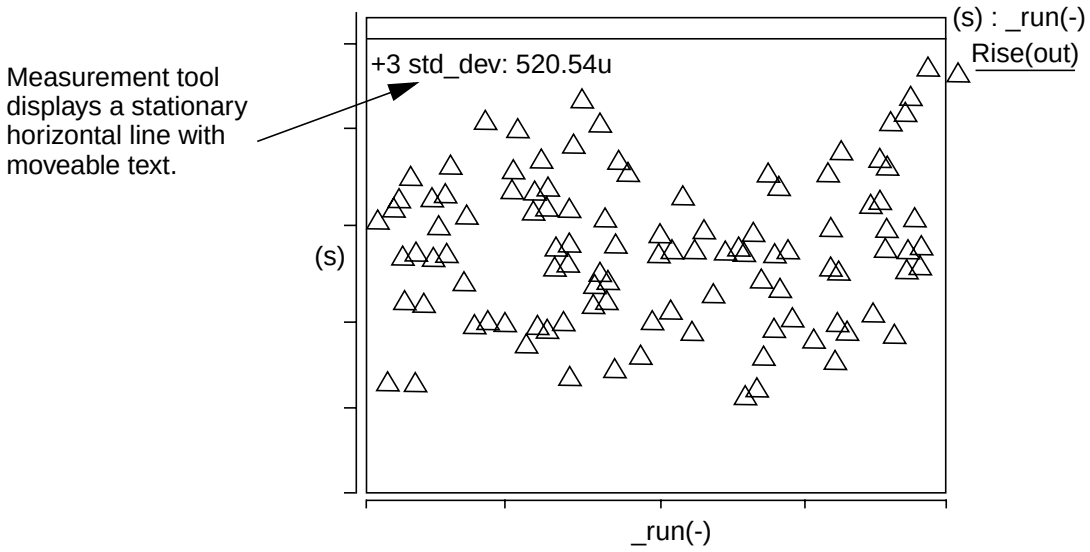
Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Mean + 3 std_dev item and any other items you want to measure.

Mean +3 std_dev Calculation

The value is calculated as mean + 3σ, where mean is the mean value and σ is the standard deviation.

Example



Mean -3 std_dev

Description

Displays the (mean – 3σ) value of a waveform.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, histogram, analog, event-driven analog

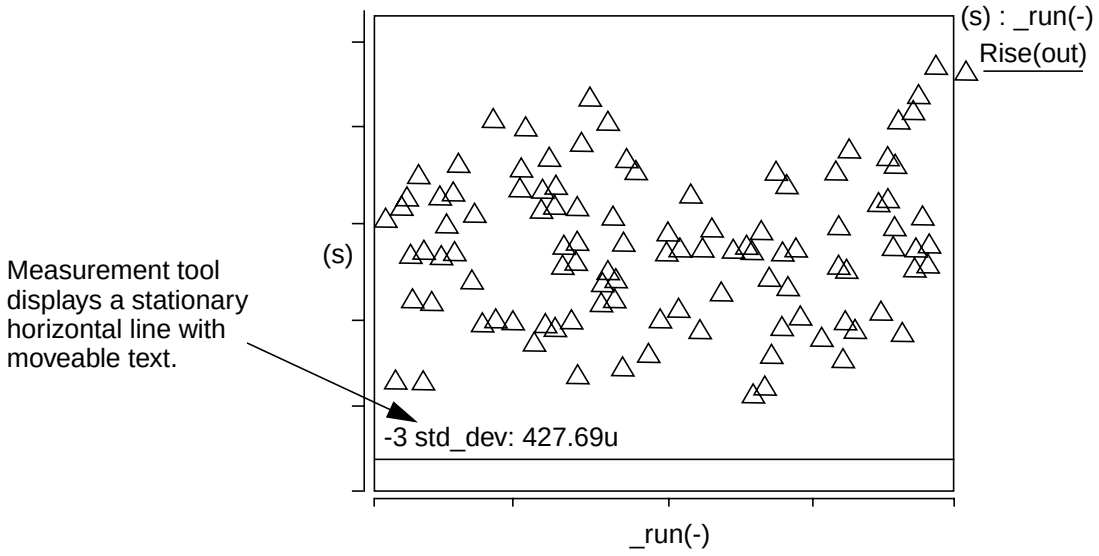
Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Mean –3 std_dev item and any other items you want to measure.

Mean -3 std_dev Calculation

The value is calculated as mean – 3σ, where mean is the mean value and σ is the standard deviation.

Example



Median

Description

Displays the median value of a waveform.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, histogram, analog, event-driven analog

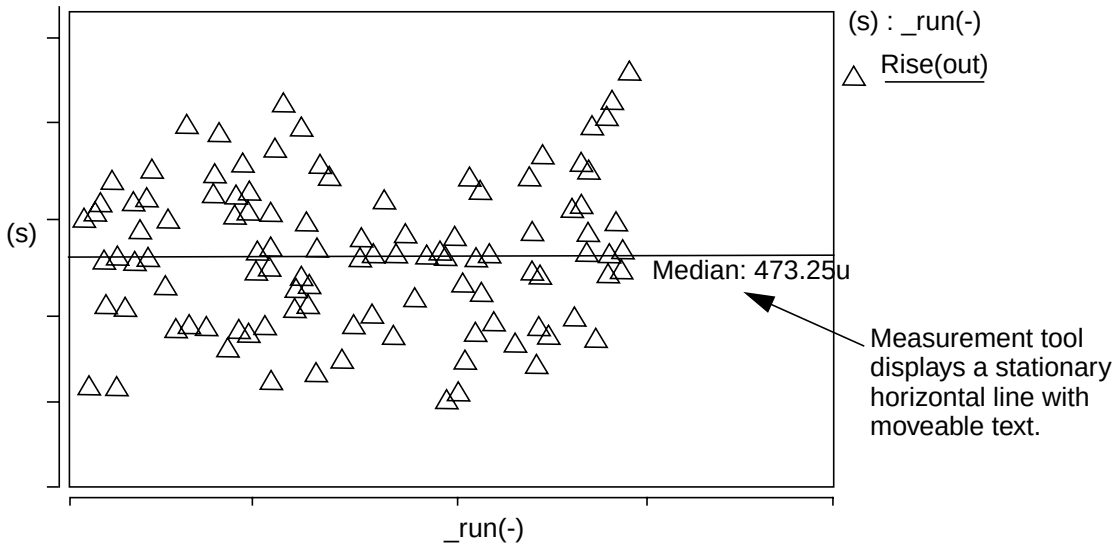
Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Median item and any other items you want to measure.

Median Calculation

The calculated median value represents the Y-axis point at which half of the data points are above and half the points are below the median value.

Example



Minimum

Description

Displays the minimum value of a waveform.

Measurement Category

Levels, Statistics

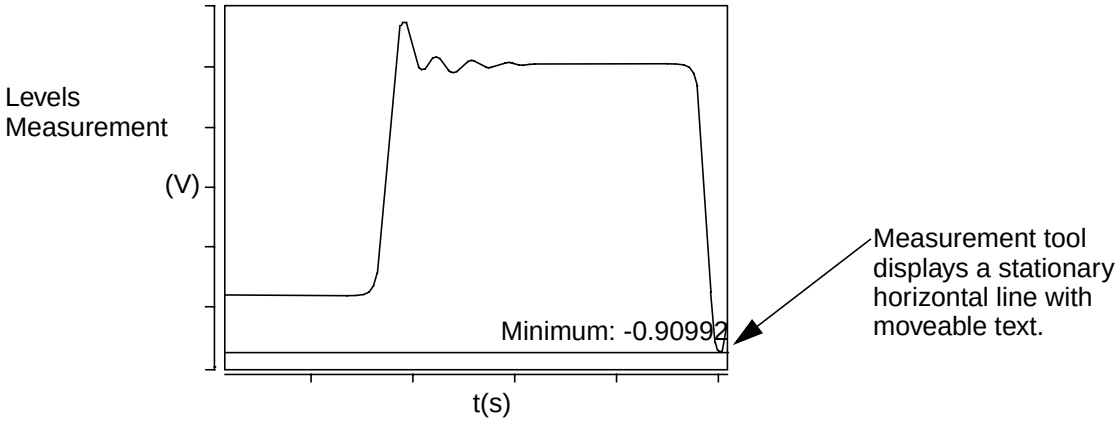
Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram, spectral

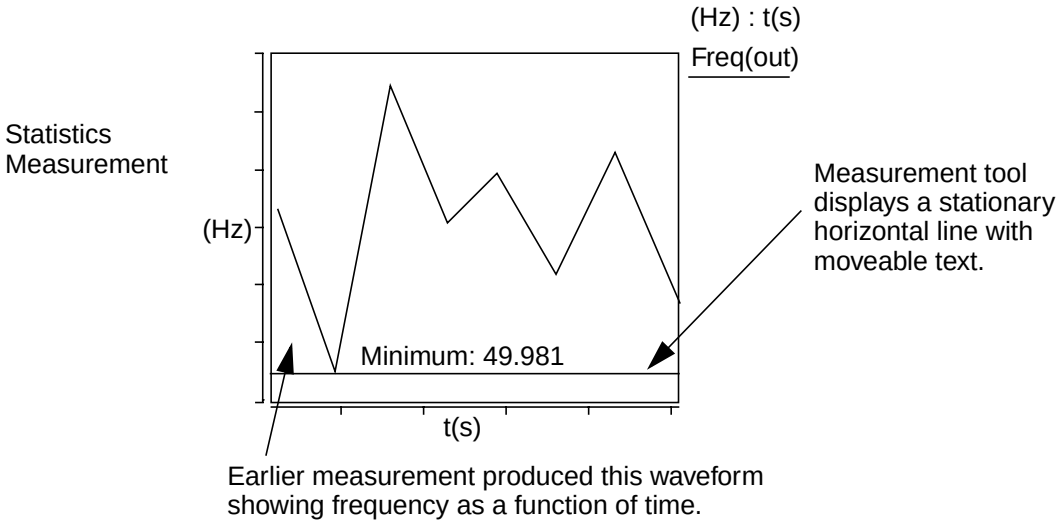
Dialog Box Fields

Category List	This measurement appears in the Statistics or Levels category. All Statistic or Levels category items appear below the Signal field (depending on which category you selected). Select the Minimum item and any other items you want to measure.

Example 1



Example 2



Natural Frequency

Description

Displays the natural frequency of a point on a waveform.

Measurement Category

s Domain

Type of Measured Waveform

Pole zero data, analog (must be complex)

Dialog Box Fields

X Value	Optional. You can provide an X-value and the tool will provide the Y-value at that coordinate. If you do not specify the X-value, a default is used.

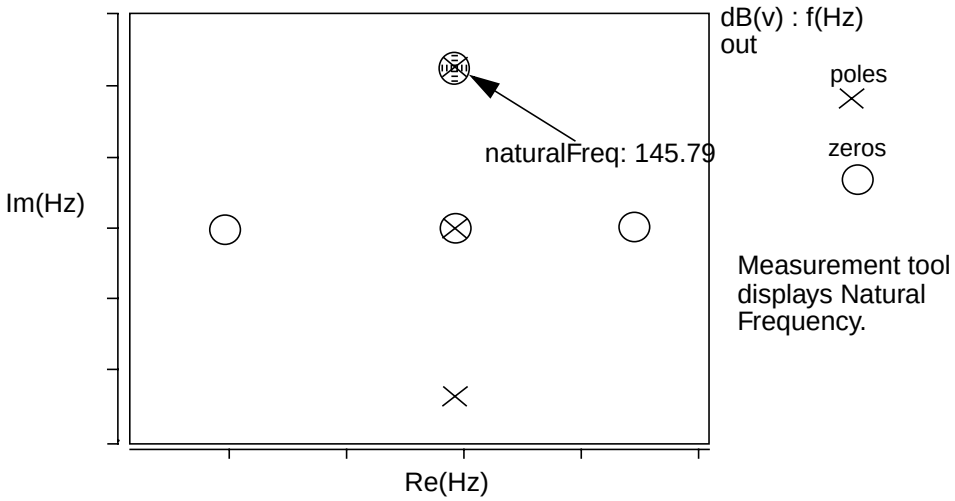
Natural Frequency Calculation

The natural frequency of a waveform is calculated as the absolute value of an argument

$$\text{natural frequency} = \sqrt{(\text{real}^2 + \text{imag}^2)}$$

.

Example



Nyquist Plot Frequency

Description

Displays the frequency at a point on a Nyquist (or Nichols) plot.

Measurement Category

Frequency Domain

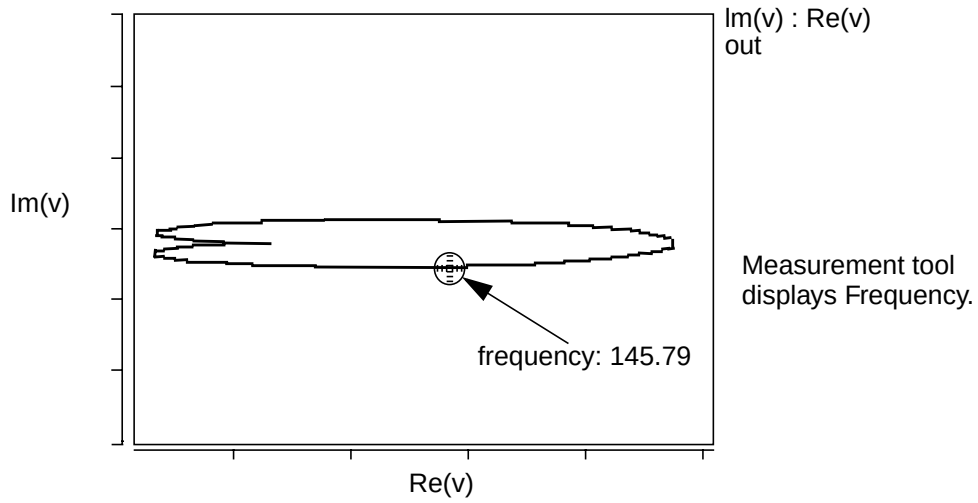
Type of Measured Waveform

Analog (must be complex)

Dialog Box Fields

X Value	Optional. You can provide an X-value and the tool will provide the frequency at that coordinate. If you do not specify the X-value, a default is used.

Example



Overshoot

Description

Displays the overshoot of a waveform relative to a default or specified topline.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Reference Levels

You can display this reference level by clicking on the Visibility Indicator at the right of the Topline field.

Topline
Specify a topline value within the upper and lower Y-axis values, or use the default value. You can display this reference level by clicking on the Visibility Indicator at the right of the Topline field.

Baseline
Specify a baseline value within the upper and lower y-axis values, or use the default value. You can display this reference level by clicking on the Visibility Indicator at the right of the Baseline field.

Measure Format

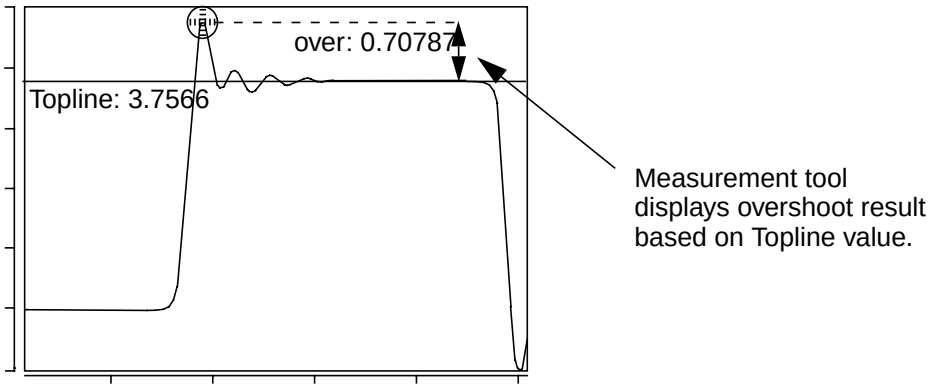
Absolute
The magnitude of the overshoot is calculated as the absolute value of an argument.

Percentage
The magnitude of the overshoot is calculated as the percentage of an argument.

Overshoot Calculation

The overshoot is calculated as the difference between the maximum point on the waveform and the specified (or calculated) Topline value. For more information on how the Topline value is calculated, refer to Topline/Baseline.

Example



Pareto

Description

Pareto Analysis is a simple method for separating the major causes (the "vital few") of a problem, from the minor ones ("trivial many").

Measurement Category

Statistics

Type of Measured Waveform

Multi-member

Dialog Box Fields

Parameter Plot File	The plot file where the parameter values were saved from the multi-member analysis. Using the downward pointing arrow allows you to use the Open Parameter File dialog box.
Parameter Names	Select All Statistical Parameters, type in individual parameters, or pick parameters with the Browse in Plot File option. Using the downward pointing arrow displays the options for this field.
R**2 and Sensitivity Histograms	Creates a new graph that contains two histograms, one labeled R**2 (the correlation coefficient) and the other labeled Sensitivity.
Scatter Plots	Generates a scatter plot for all of the important parameters showing the measured values versus the parameter values.
Save to File	Allows all of the Pareto results to be saved to a text file. By default the file name is `pareto.txt`. Using the downward pointing arrow allows you to save the file in a specific directory with a specific name.

Minimum R**2 to Display	Specify the minimum values for R**2 (the correlation coefficient). The default value is 0.1. The histograms and scatter plots will only show those parameters that have both a sensitivity value and an R**2 value greater than these values. Changing these values to 0.0 will show the effect of all of the parameters in the plot file. Depending on the design, this may result in a large number of bars in the sensitivity and R**2 histograms and a large number of scatter plots. The minimum R**2 and minimum sensitivity values do not affect the results written to the text file if that option is selected.
Minimum Sensitivity to Display	Specify the minimum values for Sensitivity. The default value is 0.1. The histograms and scatter plots will only show those parameters that have both a sensitivity value and an R**2 value greater than these values. Changing these values to 0.0 will show the effect of all of the parameters in the plot file. Depending on the design, this may result in a large number of bars in the sensitivity and R**2 histograms and a large number of scatter plots. The minimum R**2 and minimum sensitivity values do not affect the results written to the text file if that option is selected.

Pareto Calculation

The sensitivity histogram is similar in concept to the results from a Saber sensitivity analysis, except that the values from a Saber sensitivity analysis will be based on varying one parameter at a time, while the sensitivity histogram from the Pareto analysis accounts for the variation of all parameters at once. In addition, the size of the parameter change for a Saber sensitivity is a small percentage of the nominal parameter value, while the change from the Monte-Carlo analysis is related to the distribution of the parameter and is typically much larger. These are important distinctions for nonlinear circuits where two or more parameters may interact to influence the outputs being measured. A Saber sensitivity analysis cannot account for these influences.

The Pareto sensitivity values are calculated by fitting a straight line to the measured output versus each of the parameters. The normalized slope is the sensitivity value displayed in the histogram. The values in the histogram can be interpreted in the same way as the Saber sensitivity numbers. For example, if the sensitivity is -0.7 for the parameter `rnom(r.r1)` than a 1% change in `rnom(r.r1)` will lead to a -0.7% change in the measured output.

The R**2 (correlation coefficient) histogram is used in conjunction with the sensitivity histogram (which is why it is displayed on the same graph) and is a measure of the goodness of fit of the line that is used to calculate the sensitivity for each parameter. The R**2 values will always be between 0 and 1.

A value of R**2 close to 1 for a particular parameter indicates that the fit of the line is good and that there is a strong linear relationship between the measured value and that parameter, meaning that a change in that parameter will affect the measured value.

A value of R**2 close to 0 indicates that the fit is not good and that there is not a strong linear relationship between the measured value and that parameter, meaning that a change in that parameter will have little or no effect on the measured value.

The parameters with both a high sensitivity and a relatively high R**2 value are the ones that will have the most affect on the measured value. In other words, these are the parameters that should have a tighter tolerance to control the amount of variation in the output. In addition, these parameters can be used to change the value of the measured output. Answering the question of what is considered a high R**2 value is not easy since it depends on the interactions in the circuit. In practice, it is not uncommon for all parameters to have R**2 values less than 0.7. This does not mean that the parameters are not important, but is likely because two or more parameters interact to affect the measured output.

The scatter plots give more detailed information than is available from the R**2 and sensitivity histograms. Each scatter plot shows the measured values versus the parameter values. The best fit line through the data and the R**2 value are displayed. The more closely the points follow the best fit line, the higher the R**2 value. An R**2 value of 1 would mean that all of the points would be exactly on the line. Values of R**2 above approximately 0.7 will still show the scatter points to follow the tendency of the line. Low R**2 values, ones below approximately 0.2, will appear to be randomly placed on the graph. The slope of the best fit line is directly related to the sensitivity so the higher the slope, the more sensitive the measured value is to changes in this parameter. Of course, a sensitivity with a high slope is not very meaningful if the R**2 value is low.

Pareto Example

The following is a typical use case for the Pareto measure using the Saber Simulator:

1. Load a design into Saber and run a Monte-Carlo analysis. Be sure to save the parameters into a parameter file since they will be needed for Pareto.
2. Plot in CosmosScope one or more of the signals from the design.
3. Perform one or more measures on the signals. For example, you may be interested in the rise time at the output as well as the maximum power dissipation for a specific part. In this case, you would perform a rise time measure on the output signal and a maximum measure on the power dissipation signal for the device. Each of these measures will create a new scatter plot waveform in CosmosScope.
4. Pareto can be run individually on both of these resulting scatter plots. Select the Pareto measure in the **Measure** tool and then select the signal to be used for the analysis.
5. On the **Measure** tool in the field labeled Parameter Plot File, type in the name of the plot file where the parameter values were saved from the Monte-Carlo analysis. In most cases it will be easier to use the browse button to the right of the entry field and select the parameter file directly from the file browser dialog box.
6. Optionally select which parameters are to be used by the Pareto analysis. The default is to use all parameters from the parameter plot file that had a variation in the Monte-Carlo analysis. These are referred to as the statistical parameters. If you want to choose a subset of these parameters, you can type in the parameter names directly or you can use the browse button on the right side of the entry field. In almost all cases, using the default, All Statistical Parameters, is recommended.
7. Select the type of output from the measure. There are 3 choices:
   - R**2 and Sensitivity Histograms
   - Scatter Plots
   - Save to File

P1dB Measurement

Description

When the amplitude of the input signal is small enough, it is almost the same as (linear) AC analysis. but when the input signal becomes large, circuit response is saturated and the output amplitude does not linearly increase. The 1dB compression point can be calculated from HB SWEEP results. Sweep the input amplitude and measure the output amplitude.

Measurement Category

RF

Type of Measured Waveform

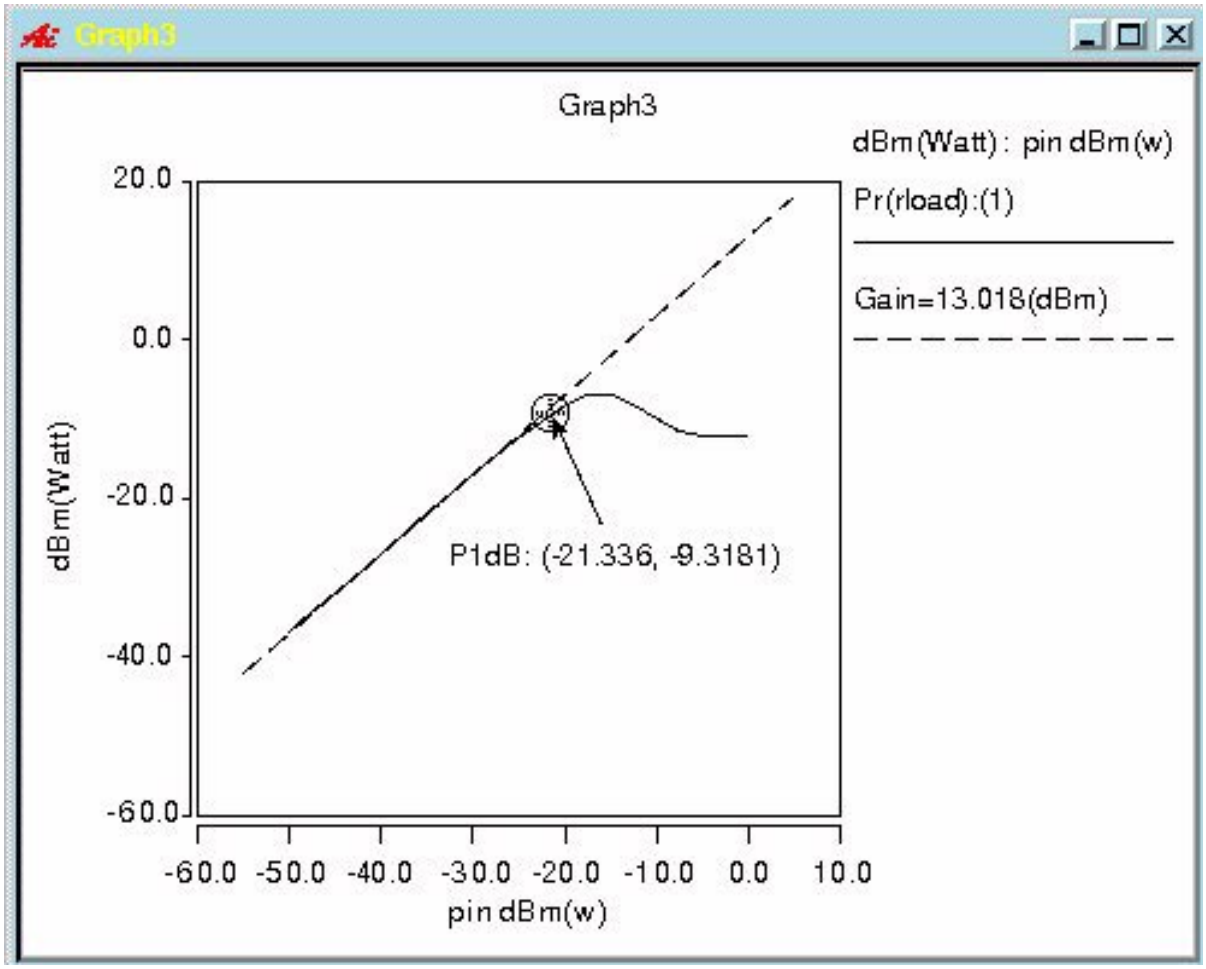
Analog, Power type, either PowerOut versus PowerIn or PowerOUt versus Frequency with PowerIn as a sweep parameter.

Dialog Box Fields

Given on waveform, PowerOut versus PowerIn, Dialog Gox Fields show:

PowerOut	PowerOut versus PowerIn curve
PowerIn	PowerIn value

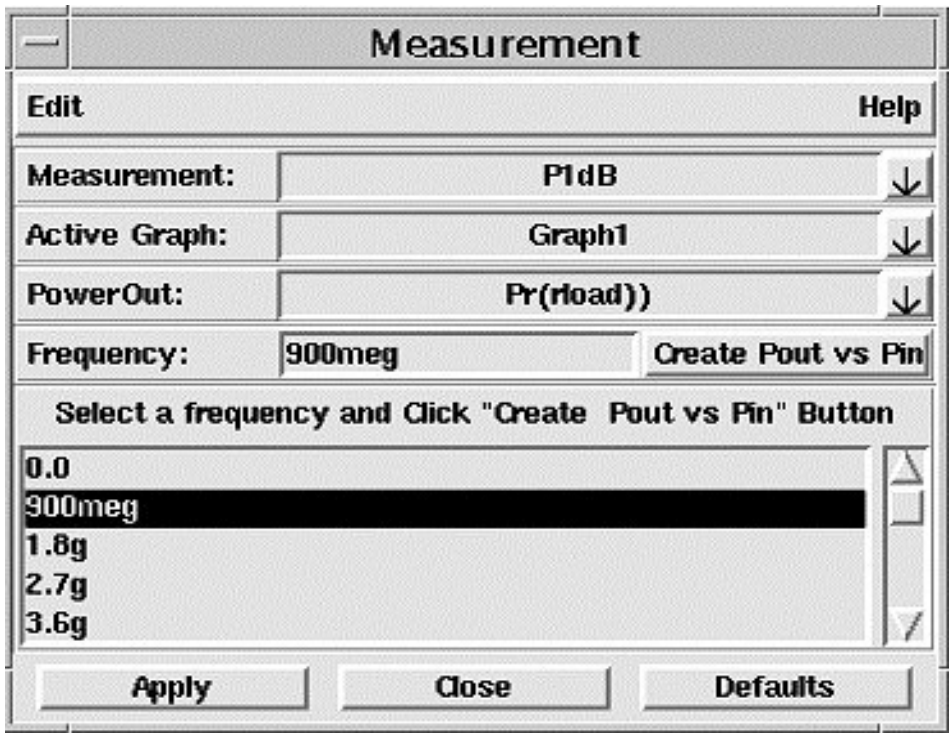
Examples

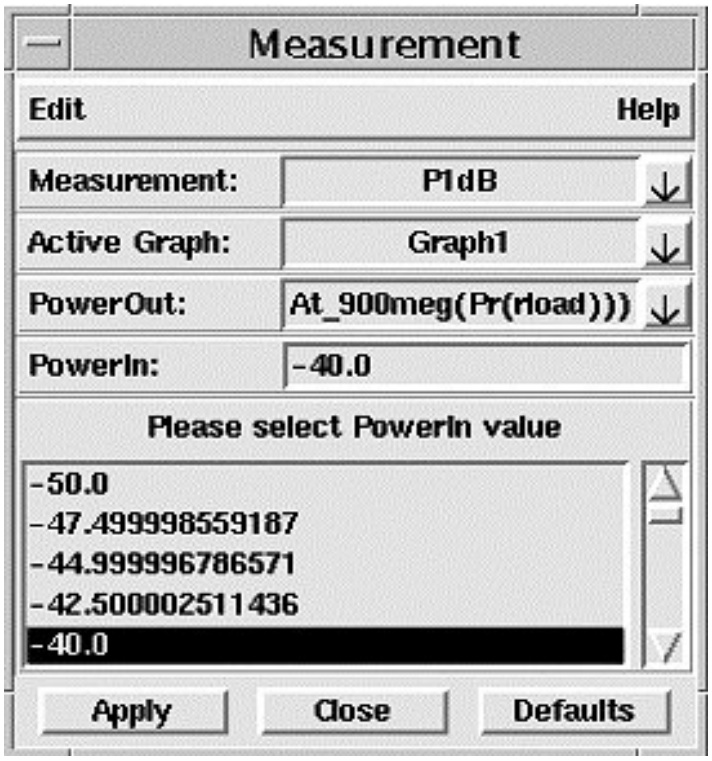


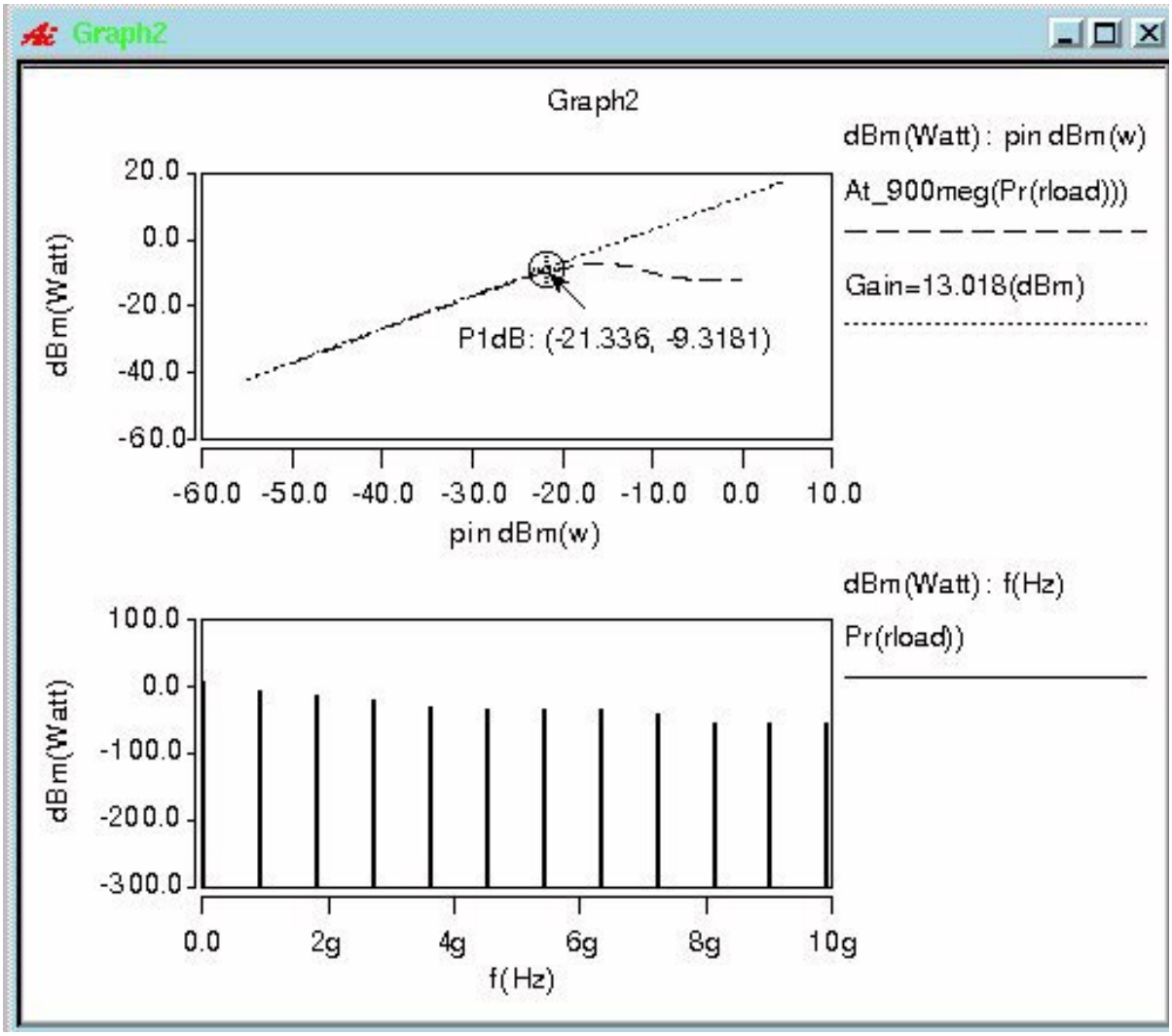
Given one waveform, PowerOut versus Frequency with PowerIn as a sweep paramter, the following Dialog Box Fields show:

PowerOut	Waveform of PowerOut versus Frequency with PowerIn as a sweep parameter
Frequency	Frequency to generate PowerOut verus PowerIn curve

Example







Peak-to-Peak

Description

Displays the waveform's peak-to-peak value.

Measurement Category

Levels

Type of Measured Waveform

*A*nalog, event-driven analog, scatter plot

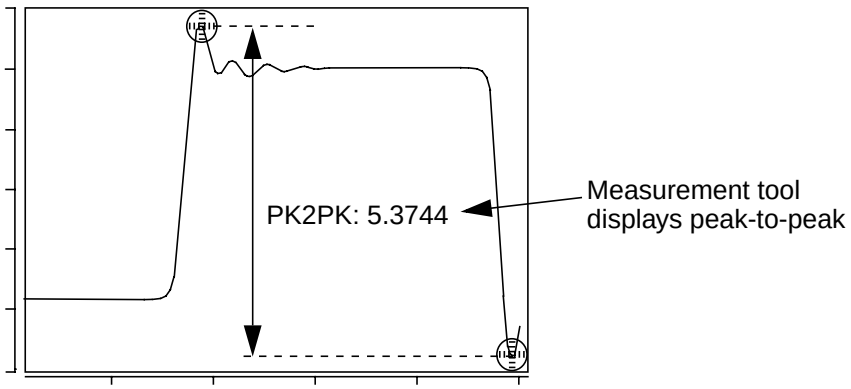
Dialog Box Fields

Category List	All `Levels` category items appear below the Signal field. Select the Peak to Peak item and any other items you want to measure.

Peak-to-Peak Calculation

The peak-to-peak value is calculated as the difference between the *maximum* and *minimum* values of the waveform.

Example



Period

Description

Displays the period of a periodic waveform relative to a default or specified topline and baseline levels.

Measurement Category

Time Domain

Type of Measured Waveform

*A*nalog, event-driven analog, digital

Possible Errors

An error is reported if the waveform does not contain at least one complete cycle.

Dialog Box Fields

Reference Levels

The following two fields set the topline and baseline levels for the measured signal and the corresponding reference signal. You can display any of these levels on the waveform by clicking on the corresponding Visibility Indicator to the right of each field.

Topline
On the measured signal, specify a topline value

Baseline
On the measured signal, specify a baseline value below the topline value, or use the default value.

Trigger

⎍⎍/⎍⎍ Specifies that the measurement starts from a period with either a rising or falling edge.

⎍⎍ Specifies that the measurement starts from a period with a rising edge.

⎍⎍ Specifies that the measurement starts from a period with a falling edge.

Create New Waveform on Active Graph or New Graph

Period vs. t - A new waveform is computed with period (y-axis) versus time (x-axis). See Example 2.

Period Calculation

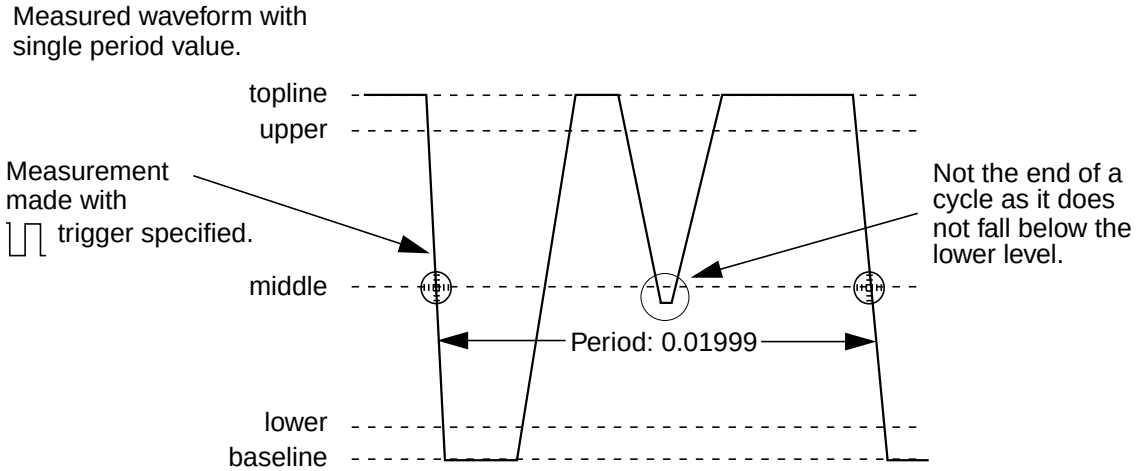
The period is calculated as the difference in time between two consecutive edges of the waveform of the same polarity as shown in the example. First, a rising or falling edge (depending on your trigger setting) is found.

A rising edge starts below the *lower* level of the waveform and rises through the *middle* level to a value above the *upper* level. A falling edge starts above the *upper* level and falls through the *middle* level to a value below the *lower* level. For more information about the *upper, middle,* and *lower* reference levels, refer to "Waveform Reference Levels".

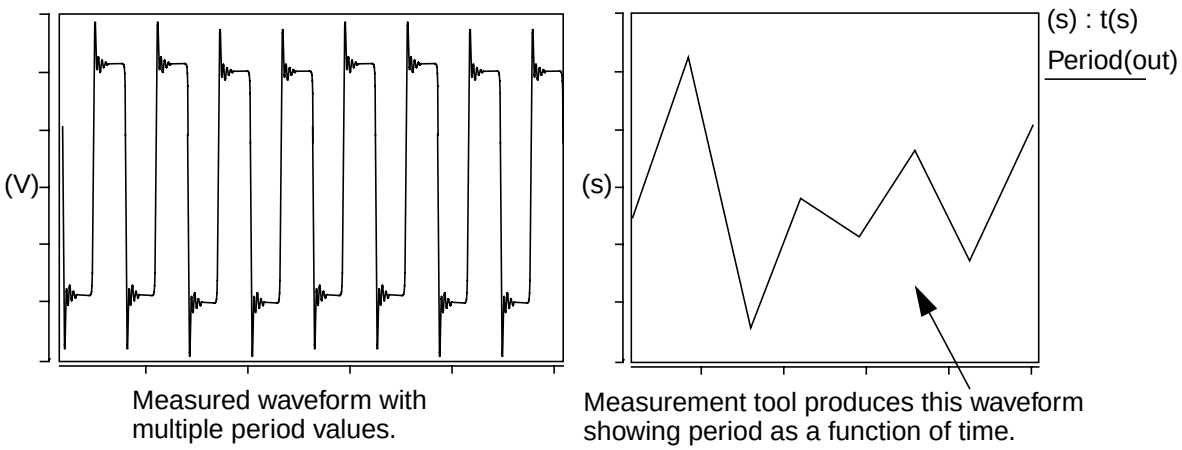
The waveform is then searched from the selected rising or falling edge to find the next edge of the same polarity. The period is calculated as the difference in time between the *middle* crossings of the two edges.

Example 1

Example 1 shows that the circled portion of the waveform is not considered an edge since it does not fall below the *lower* level of the waveform.



Example 2



Phase

Description

Displays the phase value on a point on a waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

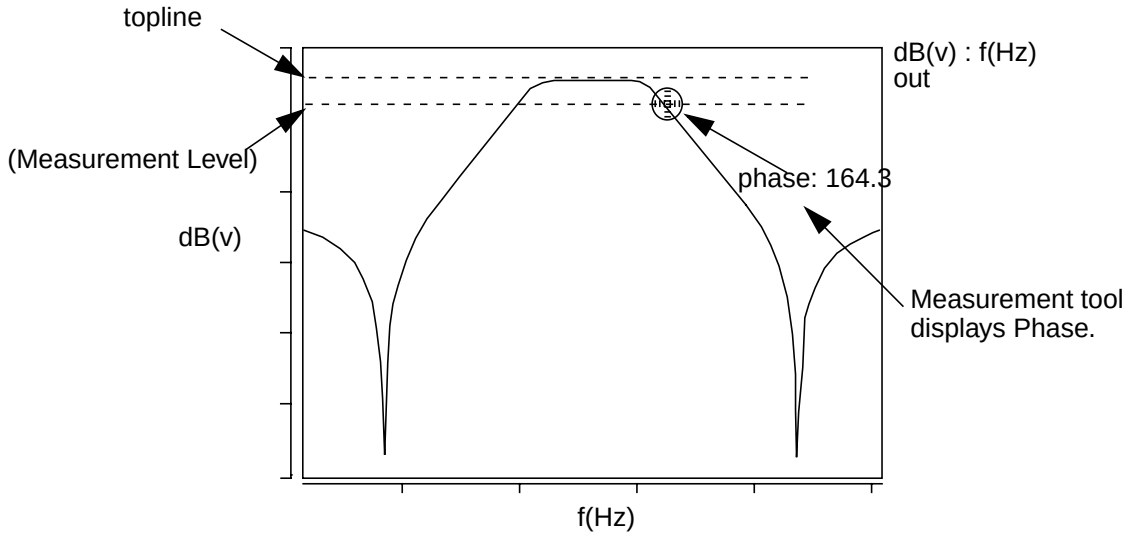
Dialog Box Fields

X Value	Optional. You can provide an x-value and the tool will provide the y-value at that coordinate. If you do not specify the x-value, a default is used.
Units	You can select radians or degrees as the units of measurement. If you do not specify, degrees is the default unit.

Phase Calculation

The phase of a waveform is calculated as atan(imag/real).

Example



Phase Margin

Description

Displays the phase margin of a complex waveform in degrees or radians.

Measurement Category

Frequency Domain

Type of Measured Waveform

*A*nalog (must be complex)

Possible Errors

An error is reported if the magnitude of the waveform does not pass through 0 dB or if the waveform is not complex.

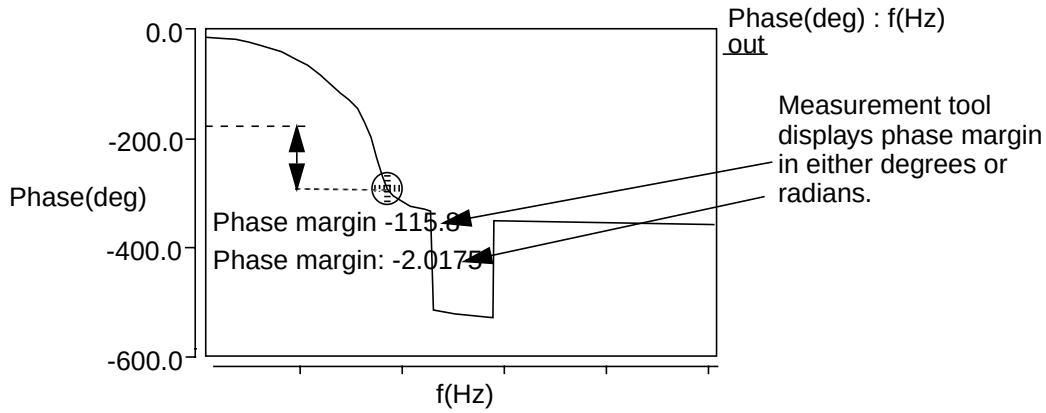
Dialog Box Fields

Units	You can select radians or degrees as the units of measurement. If you do not specify, degrees is the default unit.

Phase Margin Calculation

The phase margin is defined as the difference between the phase of the measured waveform and –180 degrees at the unity gain frequency.

Example



Point Marker

Description

Displays a moveable point marker on the waveform to display the x-value and y-value.

Measurement Category

General

Type of Measured Waveform

*A*nalog, event-driven analog, digital, scatter plot, histogram, spectral

Dialog Box Fields

Anchor Snap

Signal
Generates a moveable anchor point that snaps to the signal waveform. To see an example, refer to the Delta X Example.

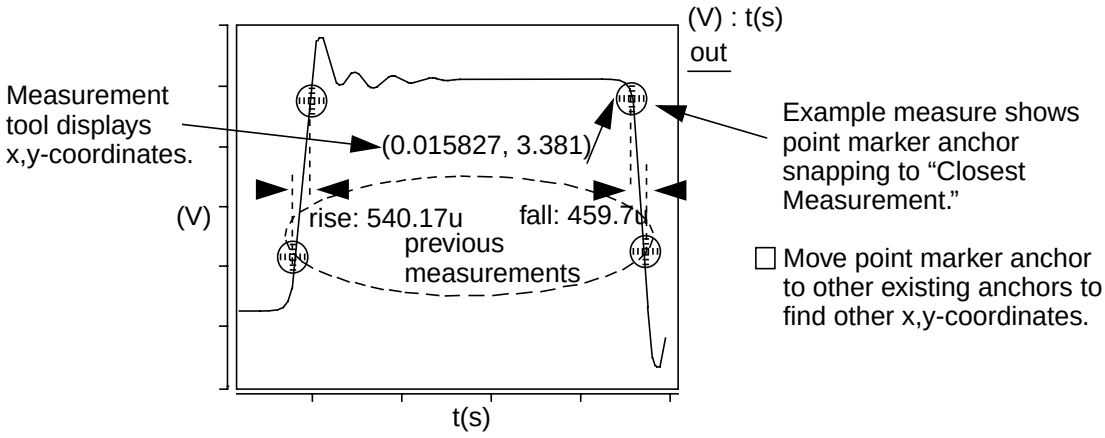
Closest Measurement
When you have multiple anchor points visible on a graph from previous measurements, this setting causes the measurement to snap to one of those nearest points. If there is no visible measurement on the active graph from which to snap, an error message is displayed. See the Example.

Floating
Generates a moveable anchor point that can be positioned anywhere within the graph. To see an example, refer to the Delta Y Example.

Location (Optional) X Value

You can optionally specify an x-value that is used to determine the corresponding y-value.

Example



Point to Point

Description

Display the following values between two X-axis points on one or two waveforms:

1. X and Y for the first point
2. X and Y for the second point
3. X-value difference between the two points
4. Y-value difference between the two points
5. Length of a straight line that connects two points
6. Slope of the two points.

If two waveforms are selected, the two waveforms do not need to be the same type, but they must be in the same graph region.

Measurement Category

General

Type of Measured Waveform

Analog, event-driven analog, digital

Dialog Box Fields

Number of Signals	Select the number of signals to apply the measurement: 1 or 2.
Signal(s)	The signals from the active graph window are listed; select the desired signal(s) to measure. When measuring 2 signals, select from the pulldown list the **Signal** and **Ref. Signal**; a **Swap** button appears to allow you to reverse the measurement operation.
Location (Optional)	Specify, if desired, two X-values used to determine the Y-values. Adjust the X1 and X2 Value entries by moving the anchor point after making the initial measurement.

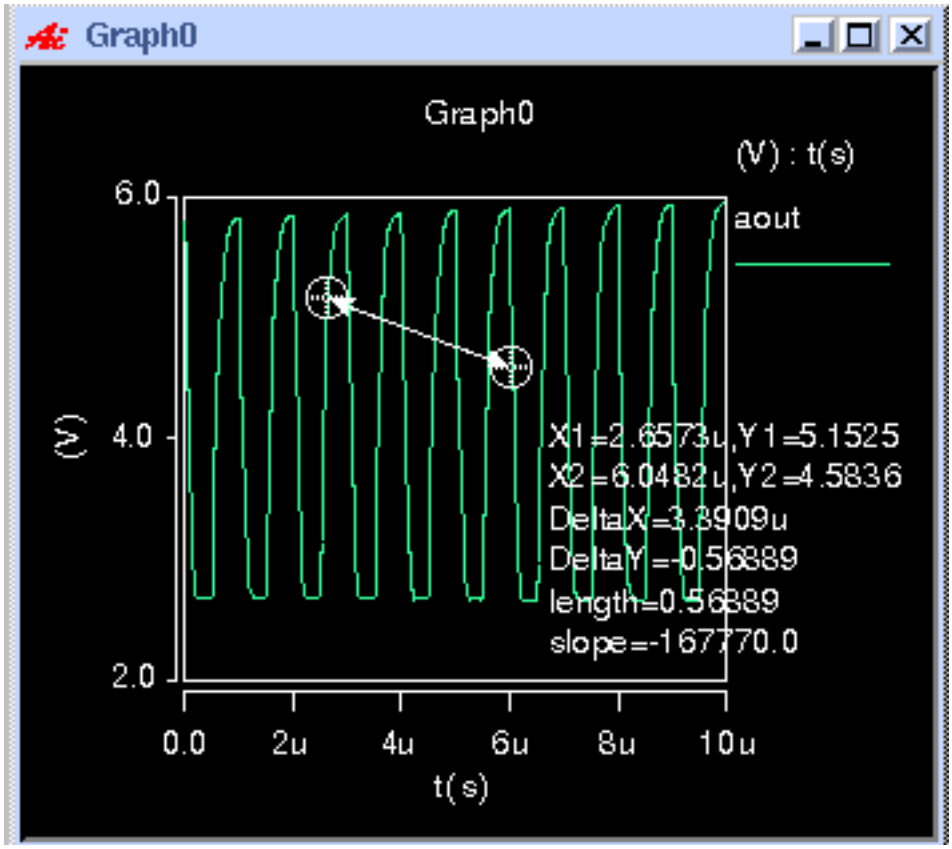
Check boxes: X1, Y1; X2, Y2; DeltaX; DeltaY; Length; Slope	Check the boxes to choose the values you want to display. All the boxes are checked by default.
Apply Measurement to:	Select from **Entire Waveform** or for the **Visible X and Y range only**.

Short Cut Icon

A short cut Icon for the Point to Point Measurement is also available:

When selecting a signal or signals from a graph region, measurement results of X and Y for the first point, X and Y for the second point, and DeltaX, DeltaY, Lenth, and Slope values.

Example



Pulse Width

Description

Displays the pulse width of a waveform relative to default or specified topline and baseline levels.

Measurement Category

Time Domain

Type of Measured Waveform

*A*nalog, event-driven analog, digital

Dialog Box Fields

Reference Levels

The following two fields set the topline and baseline levels for the measured signal and the corresponding reference signal. You can display any of these levels on the waveform by clicking on the corresponding Visibility Indicator to the right of each field.

Topline
Specify a topline value or use the default value.

Baseline
Specify a baseline value below the topline value or use the default value.

Pulse Type


Specifies that the measurement should find positive pulses.


Specifies that the measurement should find negative pulses.

Create New Waveform on Active Graph or New Graph

Pulse Width vs. t
A new waveform is computed with the pulse width values (y-axis) versus time (x-axis). See Example 2.

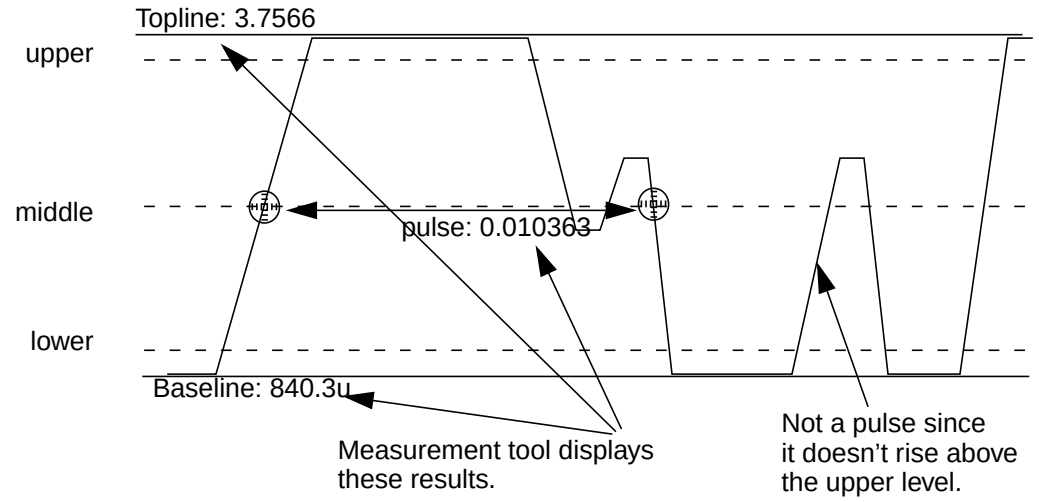
Pulse Width Calculation

To be considered for a pulse width measurement, a pulse must rise above the *upper* level and fall below the *lower* level as shown in Example 1. The pulse width is measured at the *middle* level of the waveform. For more information about reference levels of a waveform, refer to "Waveform Reference Levels".

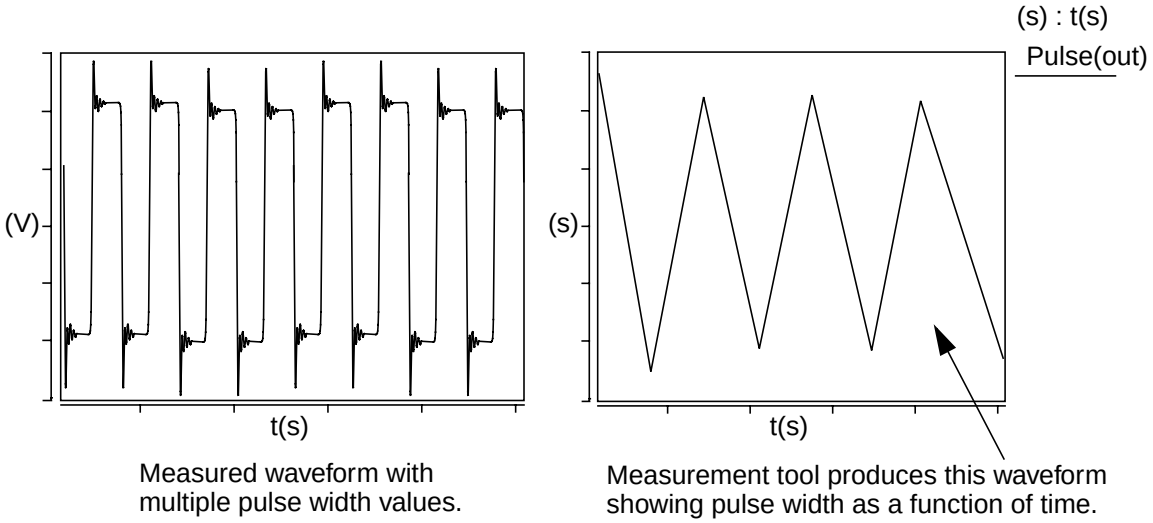
The pulse width is calculated as the difference in time between the middle level of a rising edge and the middle level of the next falling edge on the waveform.

Example 1

Measured Waveform with
Single pulse width value.



Example 2



Quality Factor

Description

Displays the quality factor of a point on a waveform.

Measurement Category

s Domain

Type of Measured Waveform

Pole zero data, complex set

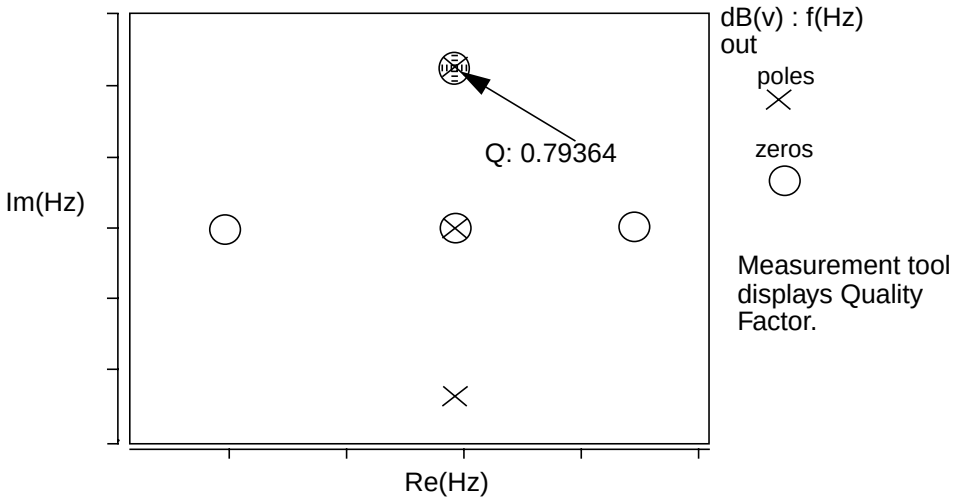
Dialog Box Fields

X Value	Optional. You can provide an x-value and the tool will provide the y-value at that coordinate. If you do not specify the x-value, a default is used.

Quality Factor Calculation

The quality factor of a waveform is calculated as 1/2(damping ratio).

Example



Range

Description

Displays the range of y-axis values covered by the waveform.

Measurement Category

Statistics

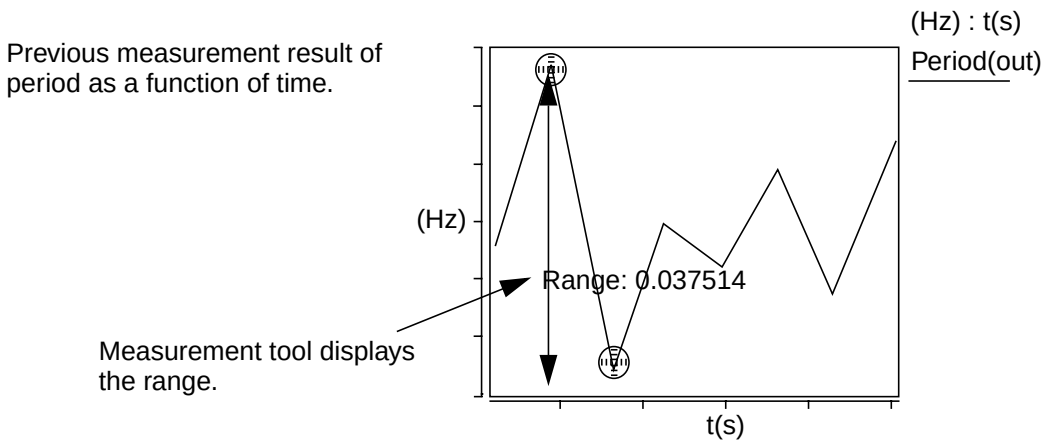
Type of Measured Waveform

Scatter plot, histogram, analog, event-driven analog

Dialog Box Fields

Category List	All Statistic category items appear below the Signal field. Select the Range item and any other items you want to measure.

Example



Real

Description

Displays the real value of a point on a waveform.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog (must be complex)

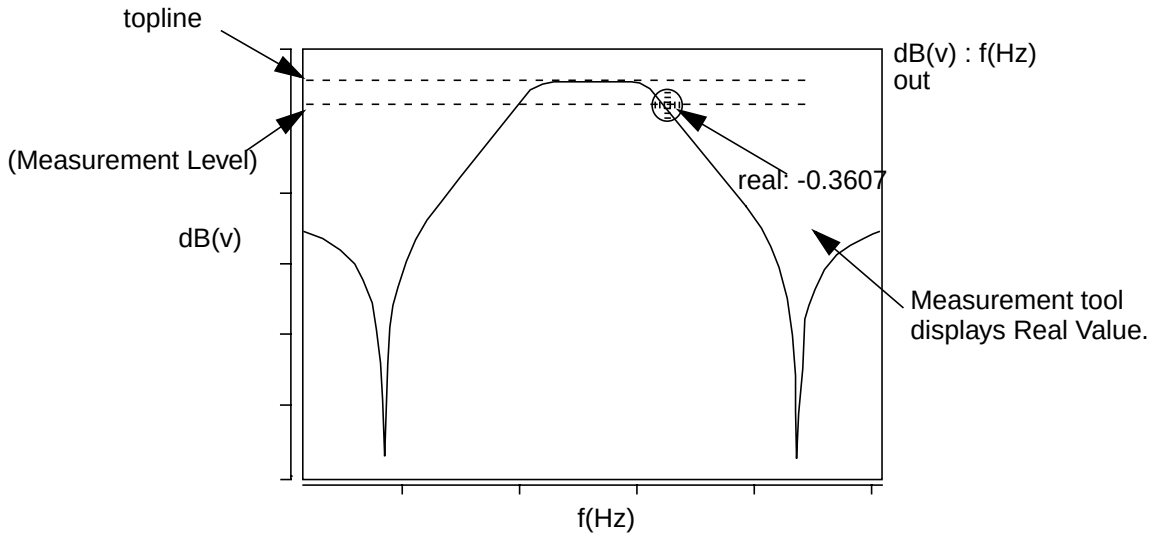
Dialog Box Fields

X Value	Optional. You can provide an x-value and the tool will provide the y-value at that coordinate. If you do not specify the x-value, a default is used.

Real Calculation

The real value of a waveform is the real part of a complex argument. If there is no real part then the value 0.0 is returned.

Example



Risetime

Description

Displays the risetime between default or selected upper and lower levels of a waveform. You can also compute the risetime based on manually-set upper/lower levels as described in the topic titled "Manually Set a Custom Topline/Baseline".

Measurement Category

Time Domain

Type of Measured Waveform

Analog

Possible Errors

An error is reported if the waveform contains no rising edges.

Dialog Box Fields

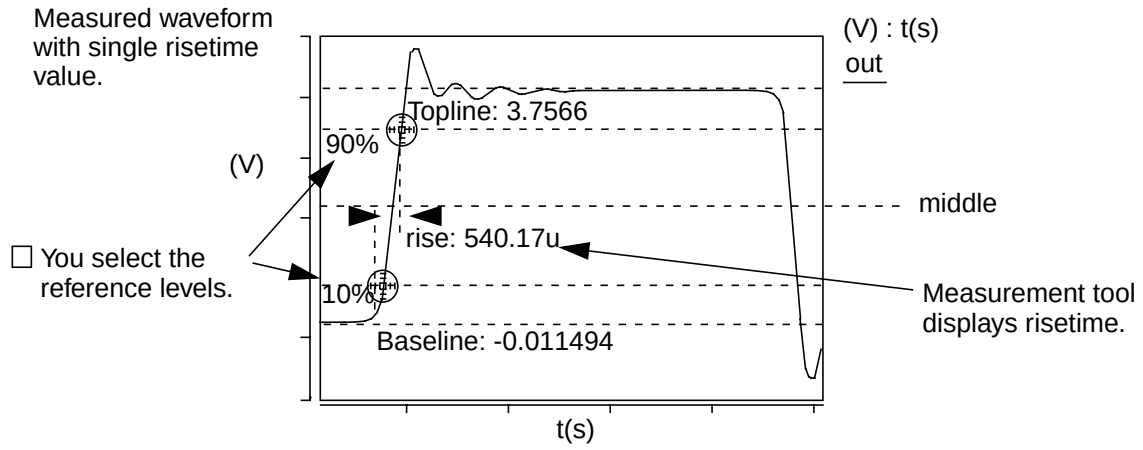
Reference Levels	The following two fields set the topline and baseline levels for the measured signal and the corresponding reference signal. You can display any of these levels on the waveform by clicking on the corresponding Visibility Indicator to the right of each field. Topline Specify a topline value or use the default value. Baseline Specify a baseline value below the topline value or use the default value.
0-100% 10-90% 20-80%	Click on one of these buttons to set an *upper* and *lower* range (in percent) relative to the topline/baseline levels. To compute a risetime based on a different percentage level than the defaults, refer to the topic titled "Manually Set a Custom Topline/Baseline".
Create New Waveform on Active Graph or New Graph	Risetime vs. t Creates a new waveform with risetime (y-axis) versus time (x-axis). See Example 2.

Risetime Calculation

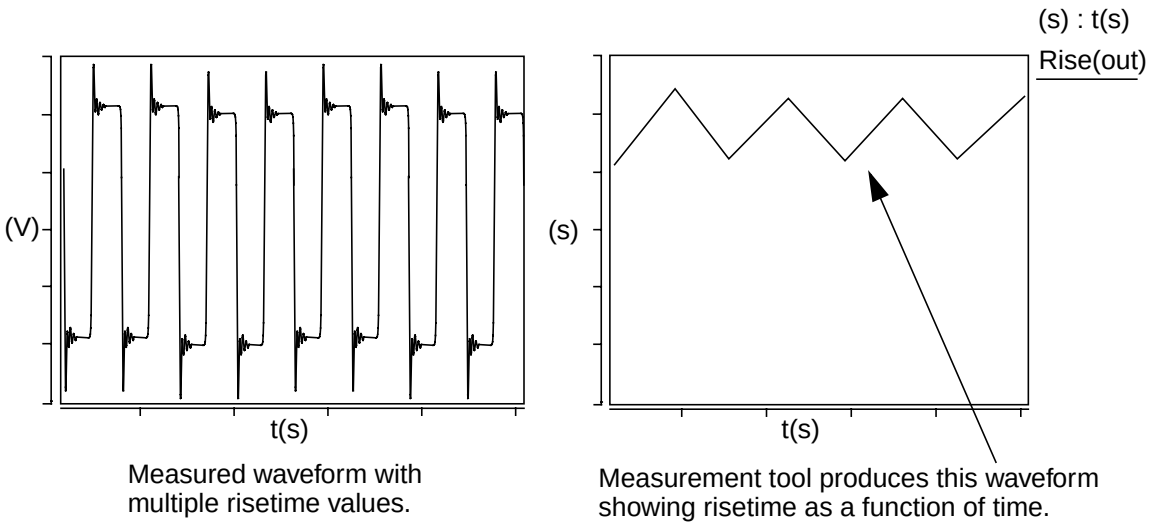
The risetime is calculated by finding a crossing with the *middle* level of the waveform. Looking forward from this point, the time when the waveform rises to the *upper* level is found. Looking backward, the time when the waveform falls to the *lower* level is found. The difference in the times is the risetime.

For more information about the *lower*, *middle*, and *upper* reference level of a waveform, refer to "Waveform Reference Levels".

Example 1



Example 2



RMS

Description

Displays the RMS value of a waveform.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram

Dialog Box Fields

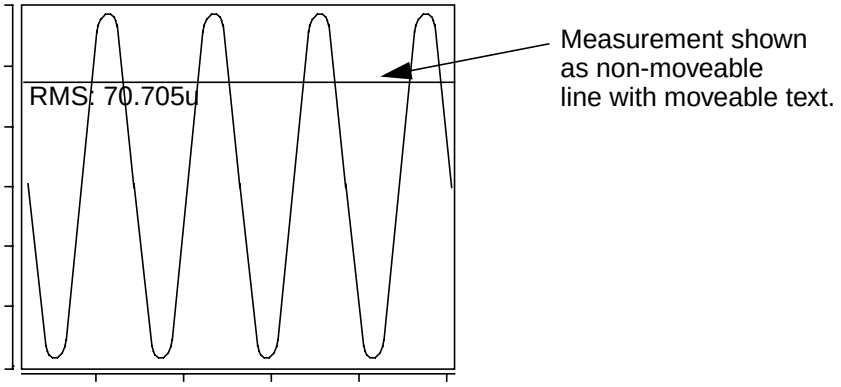
Category List	All Levels category items appear below the Signal field. Select the RMS item and any other items you want to measure.

RMS Calculation

In this calculation, `W` represents the waveform and `x1` and `x2` represent the starting and ending points.

$$\left[\frac{1}{x2-x1}\int_{x1}^{x2}(W^2 dx\right]^{1/2}$$

Example



Settle Time

Description

Displays the settle time of a waveform with respect to a default or specified settle level and a specified settle band.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Settle Level — You set a settle level or let the Measure tool calculate a default.

Settle Band — You choose the size of the settle band on either side of the settle level. The default is 5 percent of the amplitude. Other choices are as follows:

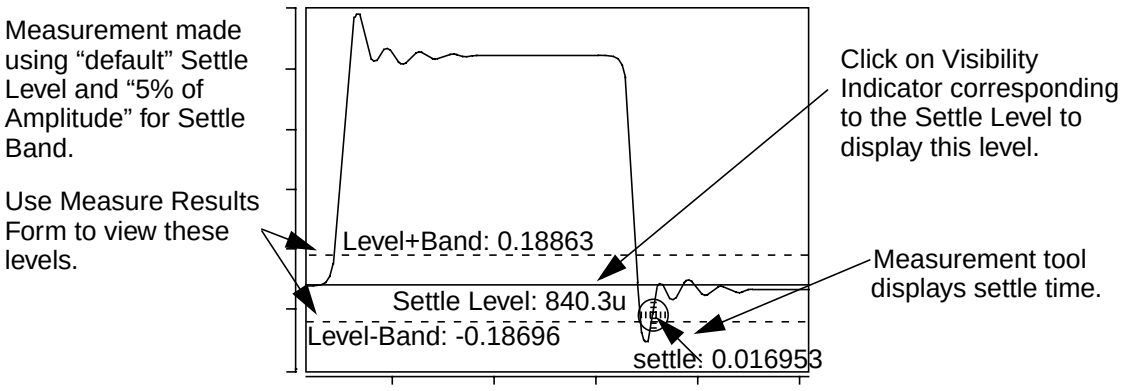
Amplitude - % of amplitude of the waveform

Settle Level - % of the settled level of the waveform

Peak to Peak - % of the peak-to-peak value of the waveform

Absolute - an absolute value such as 0.3

Example



Slew Rate

Description

Displays the slew rate of a waveform relative to default or specified topline and baseline levels.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Reference Levels

The following two fields set the topline and baseline levels for the measured signal and the corresponding reference signal. You can display any of these levels on the waveform by clicking on the corresponding Visibility Indicator to the right of each field.

Topline
Specify a topline value or use the default value.

Baseline
Specify a baseline value below the topline value or use the default value.

0-100% / 10-90% / 20-80%

Click on one of these buttons to set a range (in percent) relative to the topline/baseline levels.

Trigger

Specifies that the slew rate is calculated for rising or falling edges.

Specifies that the slew rate is only calculated for rising edges.

Specifies that the slew rate is only calculated for falling edges.

Create New Waveform on Active Graph or New Graph

Slew Rate vs. t
A new waveform is computed with slew rate (y-axis) versus time (x-axis). See Example 2.

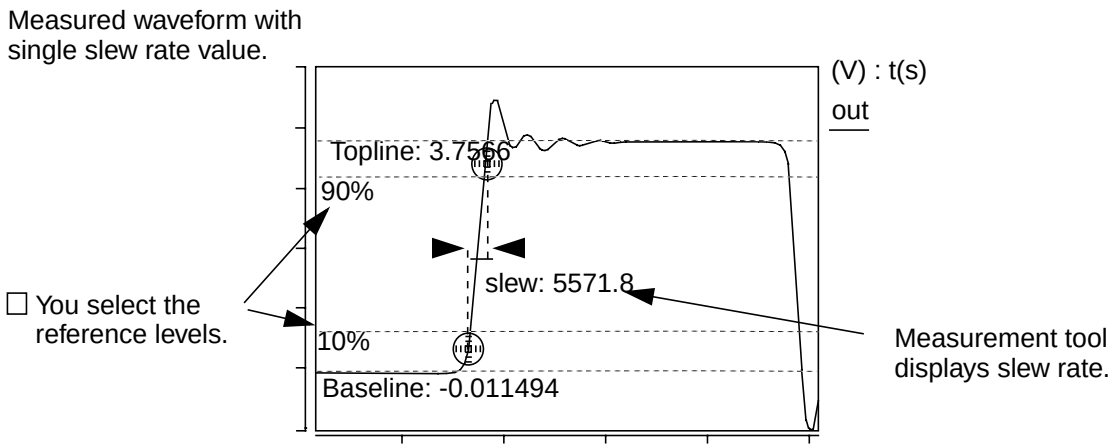
Slew Rate Calculation

The slew rate is calculated as the difference between the *upper* and *lower* levels of a waveform divided by the risetime or falltime of the edge. You select the upper and lower levels as a percent of topline/baseline.

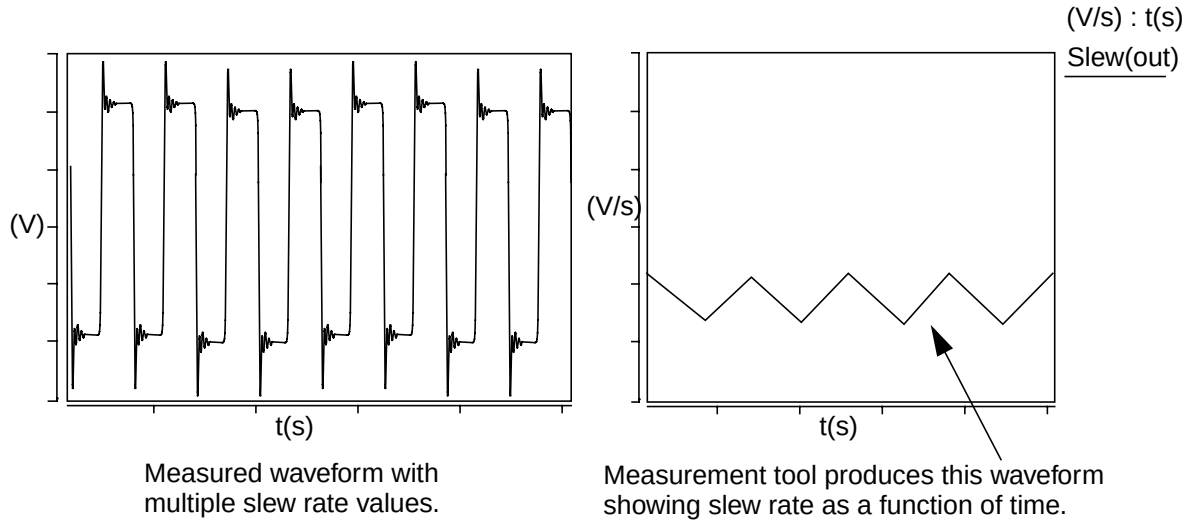
For more information about the *upper* and *lower* reference level of a waveform, refer to "Waveform Reference Levels".

For more information on how risetime is calculated, refer to the Risetime Calculation. For more information on how falltime is calculated, refer to the Falltime Calculation.

Example 1



Example 2



Slope

Description

Displays the slope (optionally as a per-octave or per-decade value) of a waveform.

Measurement Category

Frequency Domain, General

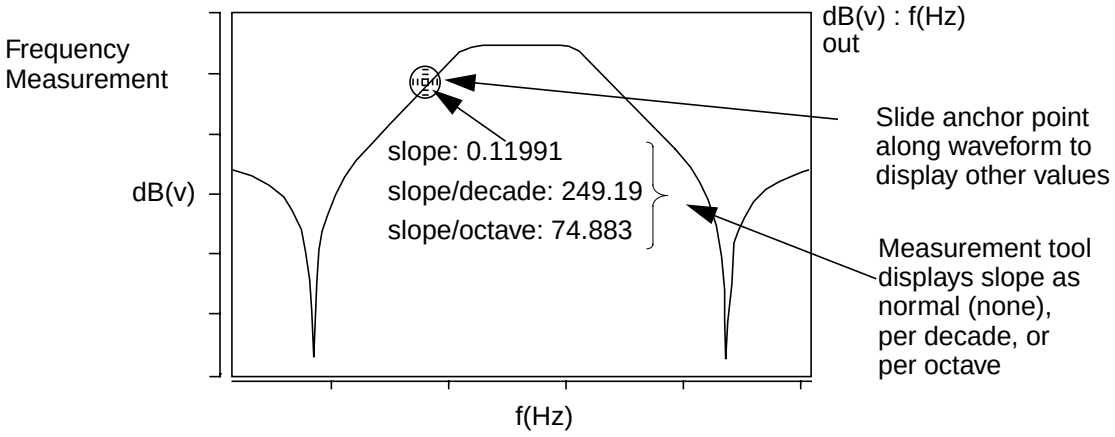
Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

X Value	Optional. You can provide an x-value and the tool will provide the slope at that coordinate. If you do not specify the x-value, a default is used.
Option (This field is visible in the Frequency Domain Category)	None Displays the slope normally. See Example. Per Decade Displays the slope per decade. See Example. Per Octave Displays the slope per octave. See Example.

Example



Standard Deviation

Description

Displays the standard deviation of a waveform. This measurement is intended for statistical (discrete) data such as histograms.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, histogram, event-driven analog

Dialog Box Fields

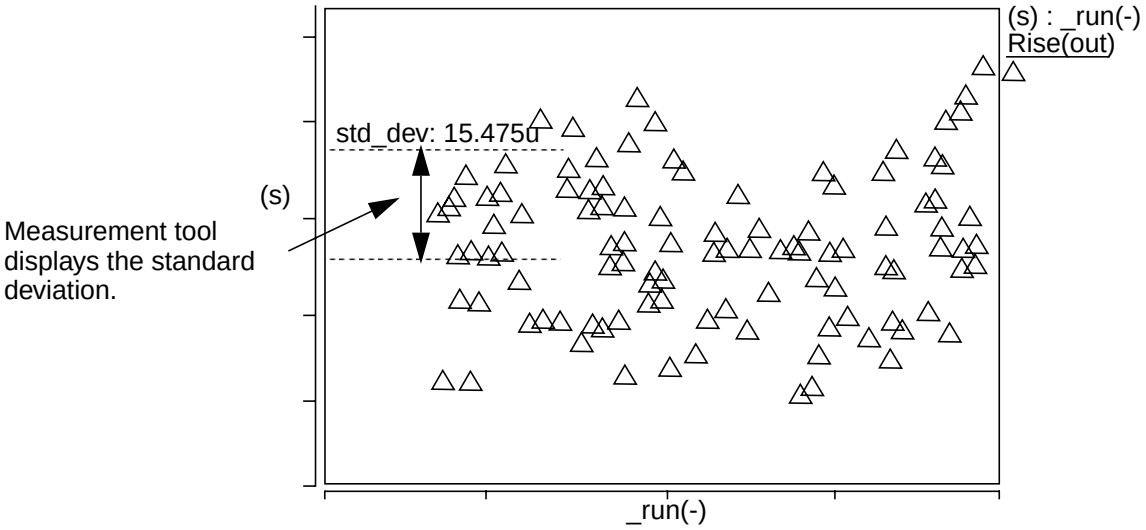
Category List	All Statistic category items appear below the Signal field. Select the Standard Deviation item and any other items you want to measure.

Standard Deviation Calculation

In this calculation, $_N$ is the number of points, W_j are the individual points of the waveform, and $_W$ is the Mean value.

$$\left[\frac{1}{N-1}\sum_{j=1}^{N}(W_j - \overline{w})^2\right]^{1/2}$$

Example



Stopband

Description

Displays the stopband, the low, high, or center frequency, or the level at which the measurement is made for a stopband-shaped waveform. The measurement is made relative to a default or specified topline level and a specified offset.

Measurement Category

Frequency Domain

Type of Measured Waveform

Analog

Dialog Box Fields

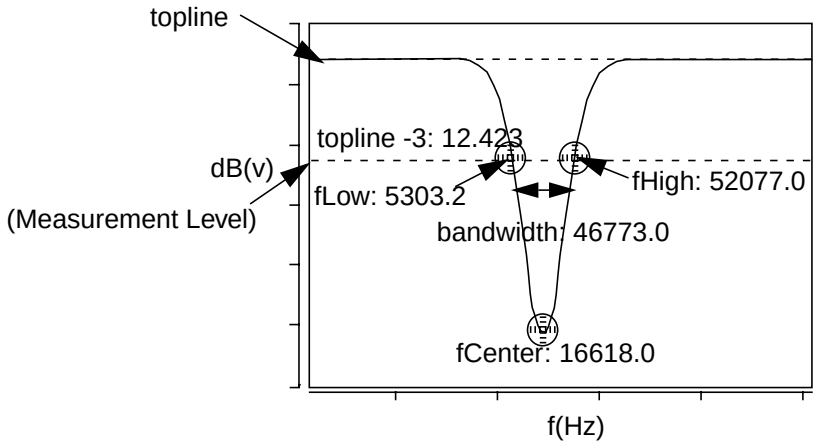
Reference Levels

If you want to see the topline and/or offset level displayed on the waveform, click on the Visibility Indicator to the right of the Topline or Offset field.

Topline
You set this field to a default or a specified level.

Offset
You specify an offset value, to be applied relative to the Topline value. The default is 3. You must also choose which operator to use (–, +, *, or /) along with the specified level. The default is the minus (–) sign. This resulting level is also called the measurement level.

Example



Threshold (at Y)

Description

Displays the x-axis values at a particular y-value on the waveform.

Measurement Category

General

Type of Measured Waveform

Analog, event-driven analog, scatter plot

Possible Errors

An error is reported if the waveform never crosses the measurement level.

Dialog Box Fields

Y value — You can optionally supply a y-value, or a default will be calculated.

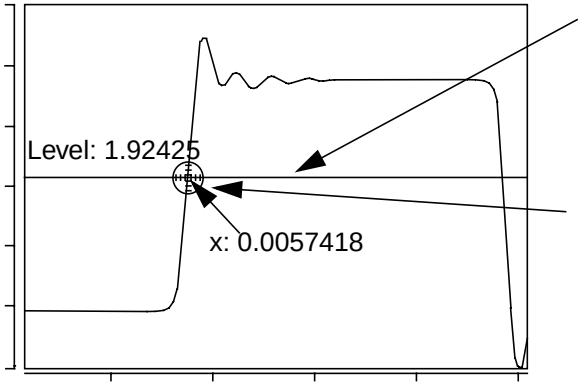
Trigger

Specifies that the slope of the waveform at the y-value can be positive or negative.

Specifies that the slope of the waveform at the y-value must be positive.

Specifies that the slope of the waveform at the y-value must be negative.

Example



Measurement tool displays a moveable horizontal line and one corresponding x-axis value at a waveform crossing.

Move anchor point along waveform to find other corresponding x values. Also use the Measure Results form to view all corresponding x values.

Topline

Description

Displays the topline level of a waveform.

Measurement Category

Levels

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Category List	All Levels category items appear below the Signal field. Select the Topline item and any other items you want to measure.

Topline Calculation

If you do not specify the topline, a default value is calculated by using a method specified in the Default Topline/Baseline field in the Measurement Preference dialog box.

Undershoot

Description

Displays the undershoot of a waveform relative to a default or specified baseline level.

Measurement Category

Time Domain

Type of Measured Waveform

Analog, event-driven analog

Dialog Box Fields

Reference Levels

Topline
Specify a topline value within the upper and lower Y-axis values, or use the default value. You can display this reference level by clicking on the Visibility Indicator at the right of the Topline field.

Baseline
Specify a baseline value within the upper and lower y-axis values, or use the default value. You can display this reference level by clicking on the Visibility Indicator at the right of the Baseline field.

Measure Format

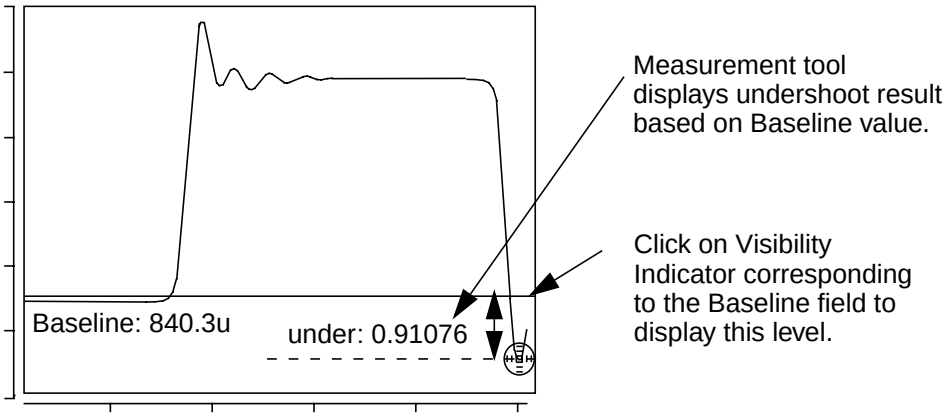
Absolute
The magnitude of the overshoot is calculated as the absolute value of an argument.

Percentage
The magnitude of the overshoot is calculated as the percentage of an argument.

Undershoot Calculation

The undershoot is calculated as the difference between the minimum point on the waveform and the specified (or calculated) Baseline value.

Example



Vertical Level

Description

Displays a moveable vertical line to identify x-axis levels.

Measurement Category

General

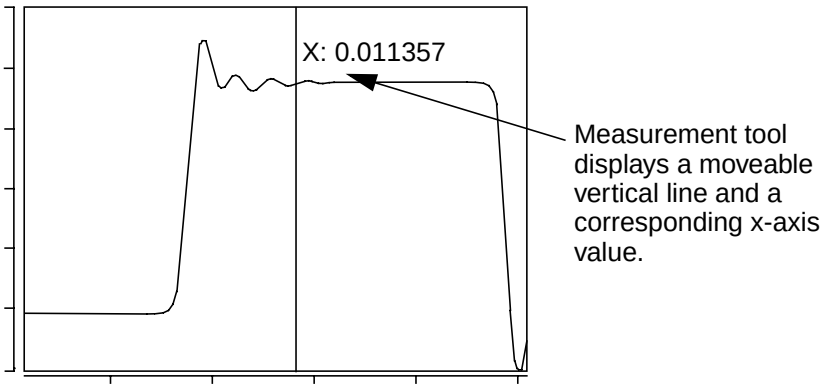
Type of Measured Waveform

Analog, event-driven analog, digital, scatter plot, histogram, spectral

Dialog Box Fields

Location (Optional) X Value	You can specify an x-value to place a vertical level on the waveform. If you do not specify one, a default value is computed.

Example



Vertical Cursor

Description

Displays a vertical cursor that spans different graphs, for X-value, Y-value, and delta Y measurements.

Measurement Category

General

Type of Measured Waveform

Any

Vertical Cursor Measurement

To access the Vertical Cursor:

1. Select **Tools > Measurement** (or select the **Measurement** button in the lower tool bar).
2. In the Measurement form, select **General > Vertical Cursor** and click the **Apply** button.

This measurement may be deleted in the Measure Results form or via a right mouse button form.

The Vertical Cursor measurement places vertical cursors in the regions of the selected signal and the reference signal, one marker in each region. You may move the marker if there are multiple Y values at that X value. You may also move the vertical cursor horizontally. The vertical cursors related to the same measurement in different regions move simultaneously.

Vertical Cursor measurement results are in two parts:

1. X-Y values of points indicated by the two markers.
2. Delta Y between the two markers.

Measurement results are displayed beside the markers and cursors. They can also be viewed in the Measure Results form.

X at Maximum

Description

Displays the x-value corresponding to the maximum value of a waveform.

Measurement Category

Levels

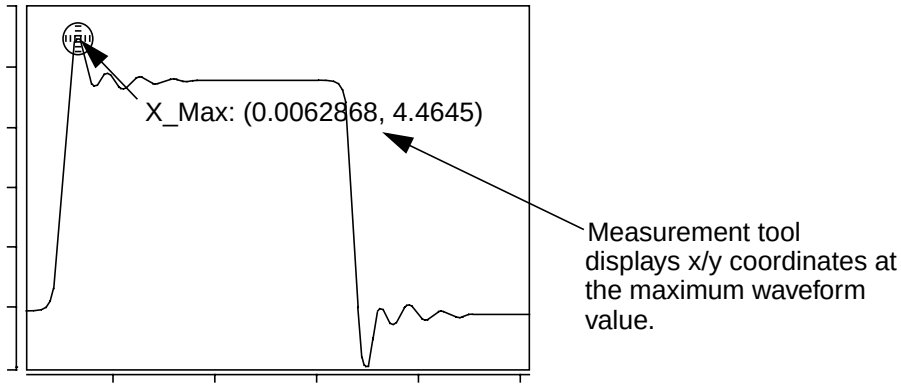
Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram, spectral

Dialog Box Fields

Category List	All Levels category items appear below the Signal field. Select the X at Maximum item and any other items you want to measure.

Example



X at Minimum

Description

Displays the x-value at the minimum value of a waveform.

Measurement Category

Levels

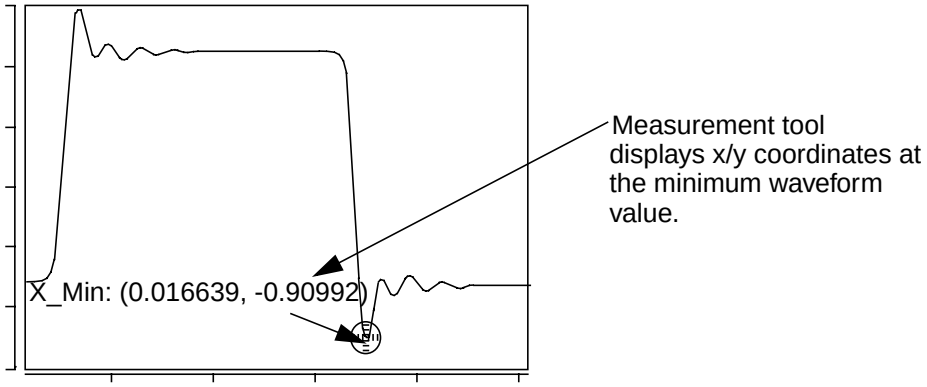
Type of Measured Waveform

Analog, event-driven analog, scatter plot, histogram, spectral

Dialog Box Fields

Category List	All Levels category items appear below the Signal field. Select the X at Minimum item and any other items you want to measure.

Example



Yield

Description

Displays the ratio of data points that fall between the specified upper and lower y-axis values of a waveform.

Measurement Category

Statistics

Type of Measured Waveform

Scatter plot, histogram, analog, event-driven analog

Dialog Box Fields

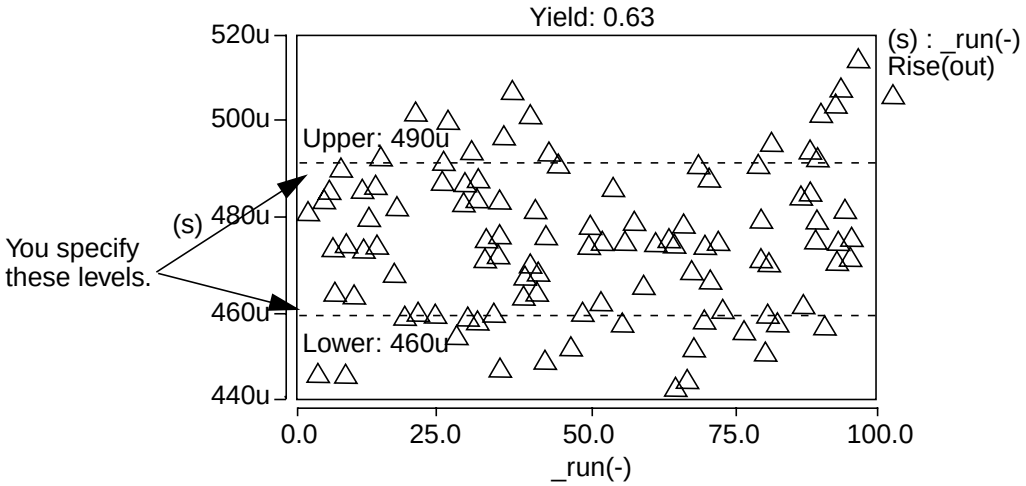
Category List	All Statistic category items appear below the Signal field. Select the Yield item and any other items you want to measure.
Specifications Limits	Required values you supply.
	Upper Specifies the upper specification limit.
	Lower Specifies the lower specification limit.

Yield Calculation

The yield is calculated as the ratio of the number of data points between the y-axis levels Upper and Lower relative to the total number of data points.

The yield results are typically only meaningful if the input waveform is uniformly spaced along the x-axis (for example, the contents of a plot file generated by a measurement operation on the results of a Monte Carlo analysis).

Example



Chapter 8

RF Tool

To perform special measurements and calculations when running RF analysis, CosmosScope provides the RF Tool in which you can select signals and apply a Point Trace measurement, or you can select a circle item and input necessary configuration information to calculate RF circles.

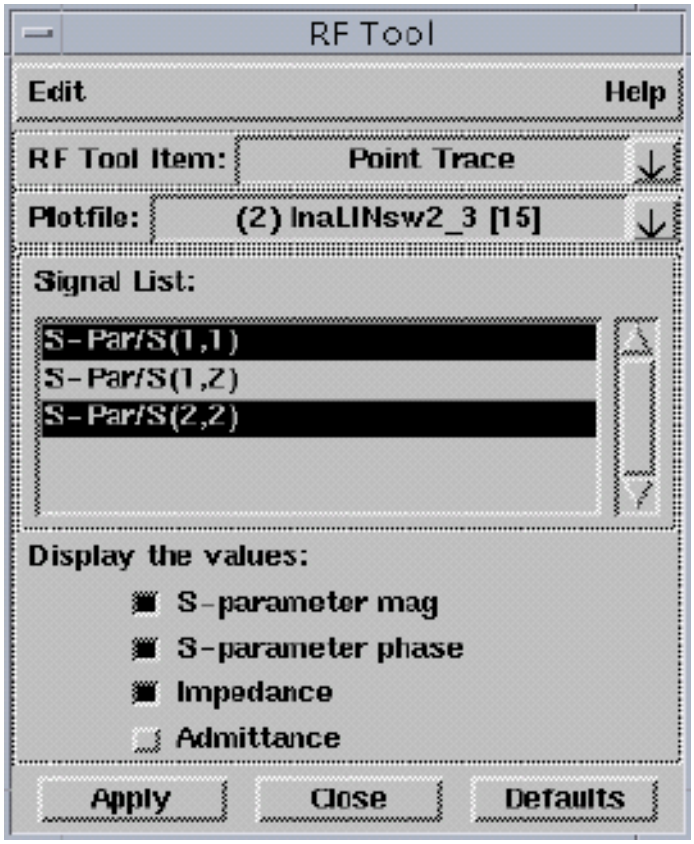
Invoking the RF Tool

1. Select **Tools > RF Tool** from the menu or click the **RF Tool** icon on the Tool Bar.
2. The RF Tool dialog box opens along with the Point Trace dialog box.
3. On the RF Tool dialog box, click the down arrow button in the Tool Item fields. You can then select the item you want to apply from the selection menu.

Point Trace Measurements

You can place Point Trace Markers for one or more signals from the same output file on Smith/Polar Charts. These markers are secured to one another; moving a marker from one chart will cause the rest of the markers to move as well. The markers will follow a trace and sequentially move from one data point to another on the same frequency value. If the signals are multi-member signals, you can make markers jump to different segments by right clicking the mouse on the marker and selecting **Next Segment** from the menu. Markers will then be displayed on the Smith/Polar Chart. A Point Trace Table will be displayed as a separate window to show the values of the markers. The results include the frequency range, the system impedance, the current frequency value, the signal name, magnitude, phase, impedance and admittance values for S-Parameters. In addition, results also include the segment index and sweep parameters for multi-member signals.

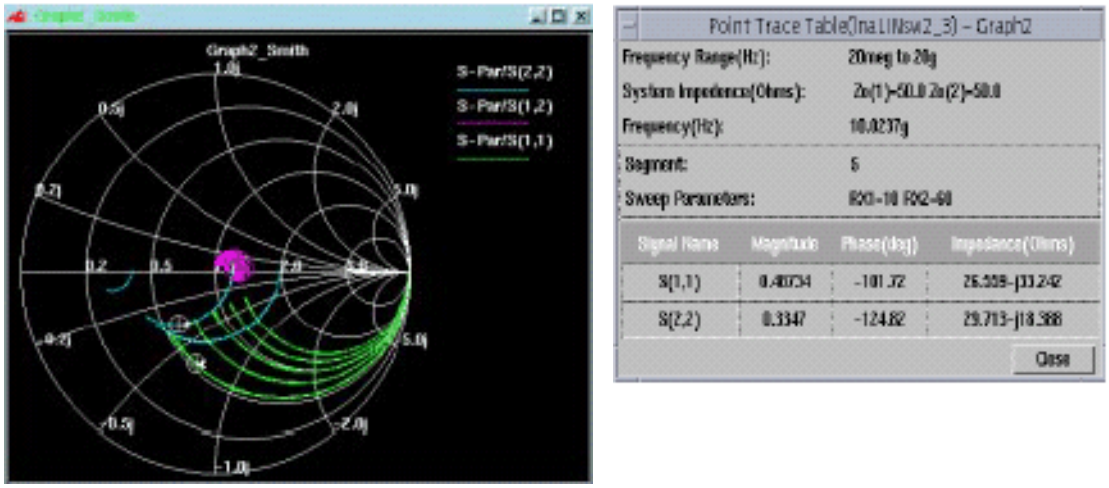
RF Tool - Point Trace dialog



Point Trace dialog

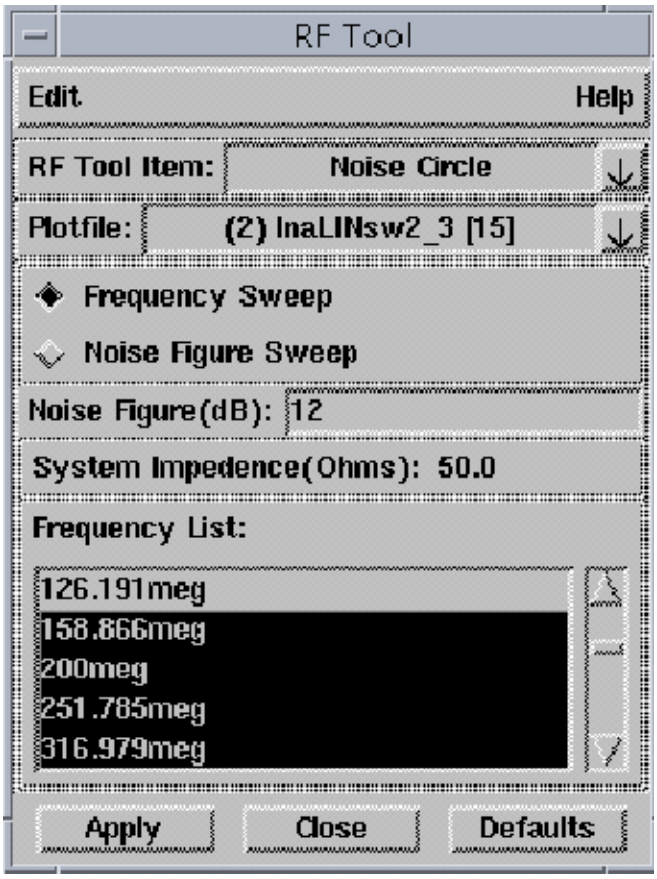
Plot File	Defines the plot file you want to use.
Signal List	Defines the signals you want to put a Point Trace on.
Display the values	Check the boxes of the values you want showing on the Point Trace table.

Point Trace Markers and Table



Point Trace Markers and Table

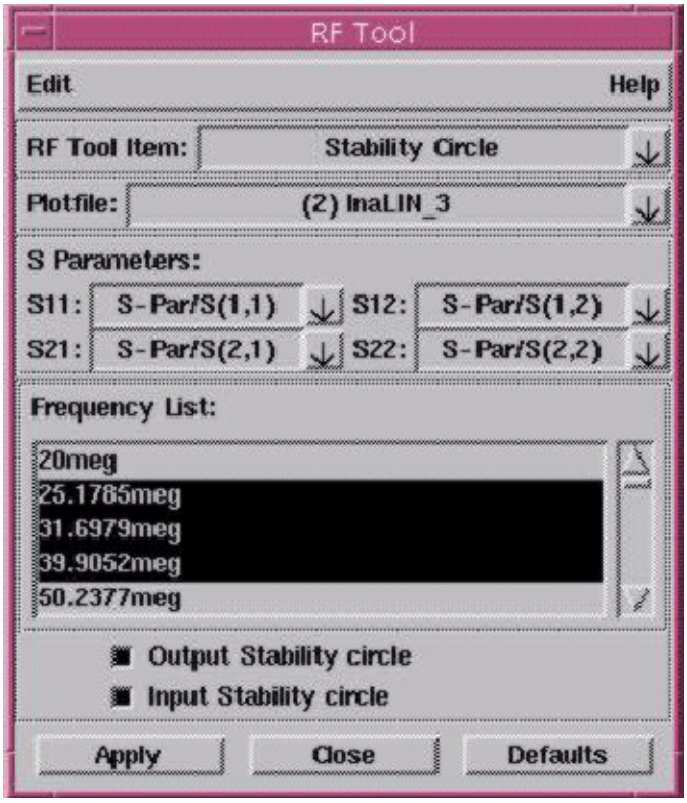
Noise Circle



Noise Circle

Plot File	Defines the plot file you want to use.
Frequency List	Defines a range of frequencies to apply to this circle. If you select **Frequency Sweep** as a sweep method, you can select multiple frequencies by holding the <Ctrl> key.
Frequency Sweep	Selects **Frequency Sweep** as sweep method for drawing the circle.
Noise Figure Sweep	Selects **Noise Figure Sweep** as sweep method for drawing the circle.
Noise Figure (db)	Defines a value for noise figure.
System Impedance (ohm)	Defines a value for system impedance.

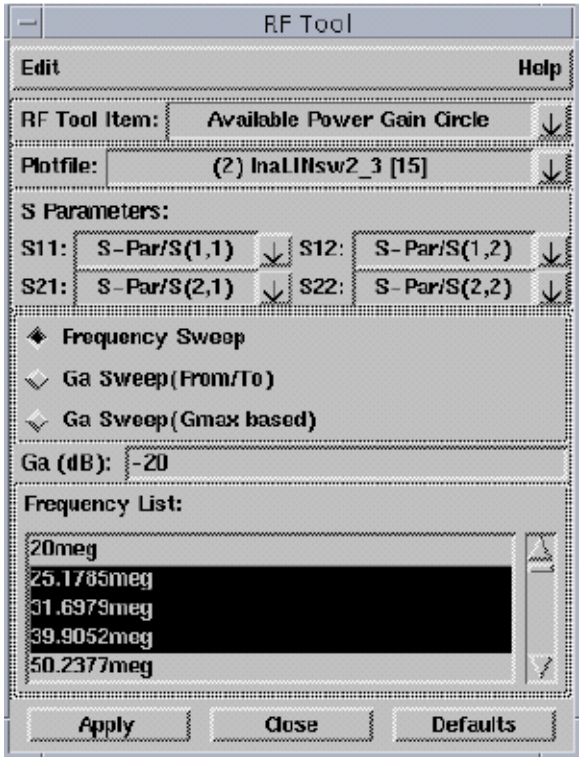
Stability Circle



Stability Circle

Plot File	Defines the plot file you want to use.
S-Parameters (S11, S12, S21, S22)	Defines the values for S-Parameters.
Frequency List	Defines a range of frequencies to apply to this circle.
Output Stability Circle	Selects the **Output Stability circle**.
Input Stability Circle	Selects the **Input Stability circle**.

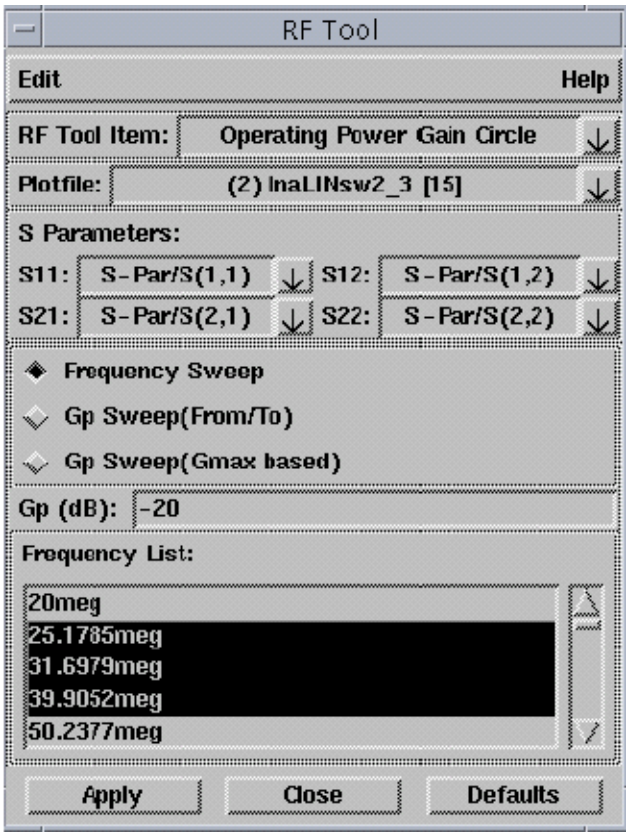
Available Power Gain Circle



Available Power Gain Circle

Plot File	Defines the plot file you want to use.
S-Parameters (S11, S12, S21, S22)	Defines the values for S-Parameters.
Frequency List	Defines a range of frequencies to apply to this circle.
Frequency Sweep	Selects **Frequency Sweep** as sweep method.
Ga Sweep (From/To)	Selects **Ga Sweep (From/To)** as sweep method.
Ga Sweep (Gmax based)	Selects **Ga Sweep (Gmax based)** as sweep method.

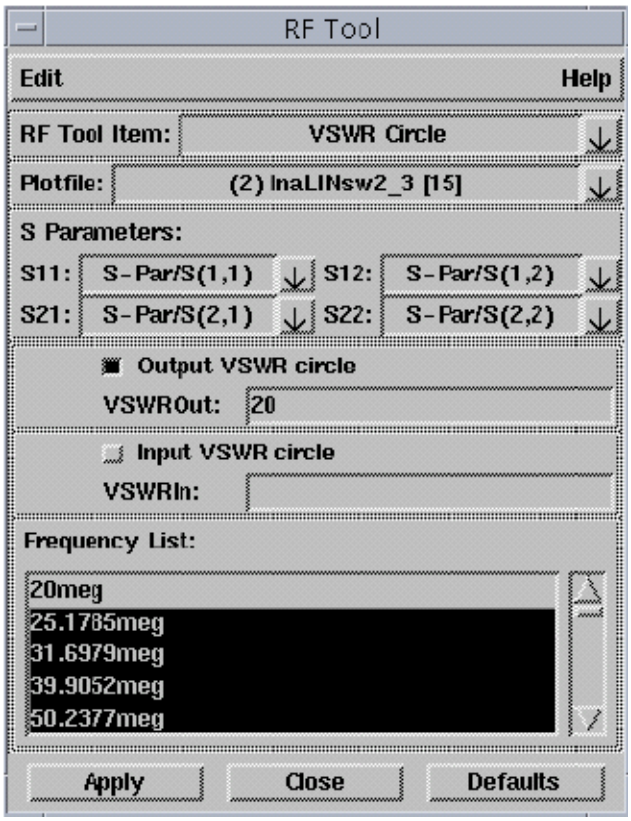
Operating Power Gain Circle



Operating Power Gain Circle

Plot File	Defines the plot file that you want to use.
S-Parameters (S11, S12, S21, S22)	Defines the values for S-Parameters.
Frequency List	Defines a range of frequencies to apply to this circle.
Frequency Sweep	Selects **Frequency Sweep** as sweep method.
Ga Sweep (From/To)	Selects **Ga Sweep (From/To)** as sweep method.
Ga Sweep (Gmax based)	Selects **Ga Sweep (Gmax based)** as sweep method.
Gp (db)	Defines the value of Gp.

VSWR Circle



VSWR Circle

Plot File	Defines the plot file you want to use.
S-Parameters (S11, S12, S21, S22)	Defines the values for S-Parameters.
Frequency List	Defines a range of frequencies to apply to this circle.
Output VSWR Circle	Selects the output VSWR circle.
VSWROut	Defines a value for VSWROut.
Input VSWR Circle	Selects the input VSWR circle.
VSWRIn	Defines a value for VSWRIn.

The RF Tool will find the associated waveforms from the selected plot file by default. To change the default signal, pull down the list box and select one from the Signal List dialog box.

Parameter Conversion

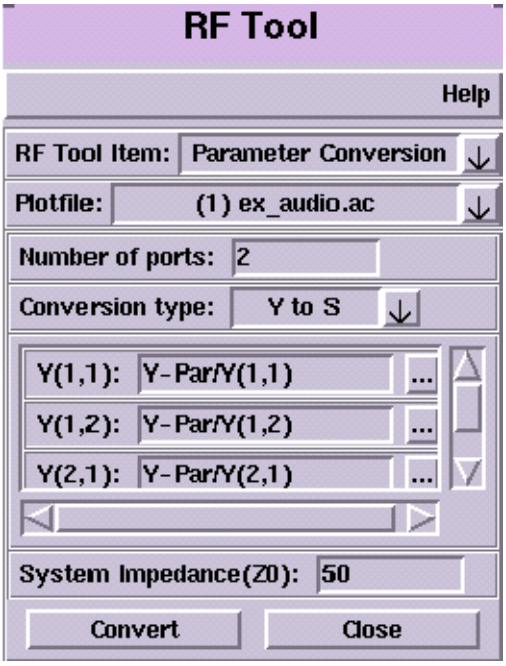
Since only S parameters are meaningful on Smith/Polar Charts and, typically, plot information is only described in terms of Y or Z parameters, the RF Tool allows for the conversion of parameters.

Conversion Procedure

In order to convert parameters:

1. Select **Parameter Conversion** in the RF Tool Item field of the RF Tool.
2. Select the plot file from the Plotfile pulldown.
3. Enter the number of ports in the Number of Ports field, with a minimum of one port.
4. Choose a conversion type from the menu, which includes S to Y, S to Z, Y to S, Y to Z, Z to S and Z to Y.
5. Set the signals for conversion.
6. Enter the system impedance (Z0) value in the System Impedance field.

   The RF Tool dialog box will look something like this:



7. Click the **Convert** button to do the conversion.

   The conversion results are displayed in a separate window containing a

list of converted signals. You can select and plot any of the signals in the list. You may also select and save any of the signals to a `*.ai_pl` plot file.

Conversion Equations

The equations used in the conversion are as follows:

```
[Y0]=[Z0]**(-1)

[Z]={{[1]+[S]}*{[1]-[S]}**(-1)}*[Z0]
[Y]={{[1]-[S]}*{[1]+[S]}**(-1)}*[Y0]
```

where

`S` are the S parameters

`Y` are the Y parameters

`Z` are the Z parameters

`Z0` is the system impedance.

Chapter 9

CosmosScope Quick Reference

This chapter provides CosmosScope reference material for the following topics:

- Icon Bar Icons
- Tool Bar Icons
- Mouse Usage
- Hot Keys (Shortcuts)
- File Types Supported by CosmosScope

Icon Bar Icons

The following lists all CosmosScope Icon Bar icons and describes where to find information on their functions.

XY Graph

Open a new, empty XY graph on the work surface.

Smith Chart

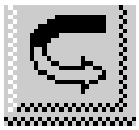
Open a new, empty Smith Chart graph on the work surface.

Polar Chart

Open a new, empty Polar Chart graph on the work surface.

Open File

Open a file.



Reload

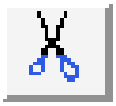
Reloads plotfiles and waveforms.

Save

Save the active window.

Print Active Window

Print hardcopy of the current window.



Cut

Copy to the clipboard and delete the original if possible.

Copy

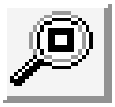
Copy to the clipboard and leave the original.

Paste

Paste from the clipboard to the active window

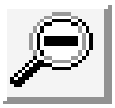
Zoom In

Increases magnification to show increased detail.



Zoom to Fit

Display all of the objects in a window.

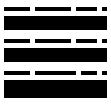


Zoom Out

Reduce magnification to show less detail, but more information.

Cascade

Place windows so that they are placed on top of each other diagonally from the upper left hand corner of the work surface to the lower right hand corner of the work surface.

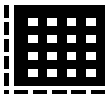


Tile

Place windows so that they do not overlap.

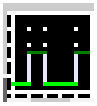
Clear Graph

Clear all signals, axes, legends, and graphics from the graph window without saving the information.



Toggle Grid

Toggle visibility of the grid on the active window.



Toggle Signal Grid

Toggle visibility of the grid on the selected digital signals.

Configure Dynamic Waveform Display

Popup a form to configure Dynamic Waveform display settings.

Create Bus

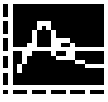
Take multiple signals and combine them into a single digital bus. If analog signals are detected, window appears requesting low and high threshold values.

Burst Bus

Expand a bus into discrete digital signals.

At X Measurement

Perform At X measurement on selected signals. For details, refer to the `Measuremen Tool` documentation.



At Y Measurement

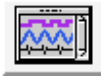
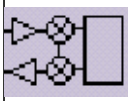
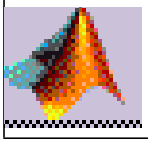
Perform Threshold (At Y measurement) on selected signals. For details, refer to the `Measurement Tool` documentation.

Point to Point Measurement

Perform Point to Point Measurement on selected signals.

Tool Bar Icons

The following table lists Tool Bar icons and describes where to find information on their functions.

Tool Icon	Function
Signal Manager 	Opens the Signal Manager. This tool allows you to browse plot files and display them as graphs.
Measurement Tool 	Opens the Measurement Tool. This tool performs measurement operations that are used to display one or more “performance measures” for a waveform. For details, refer to the Measurement Tool documentation.
Macro Recorder 	Opens the Macro Recorder Tool. This tool records a series of actions, allows you to edit these actions, and plays them back as a script.
AimDraw Tool 	Opens the AimDraw Tool. This tool allows you to create symbols in Sketch, as well as general graphic objects and text in Saber windows.
Calculator 	Opens the Calculator. This tool allows you to perform mathematical and logical operations on AIM language expressions and waveforms as well as on numbers.
Command Line Tool 	Opens the Command Line Tool. This tool allows you to enter AIM commands, write scripts, and save them into files.
RF Tool 	Opens the RF Tool. This tool allows you to perform measurements and calculations on RF simulation output files.
MATLAB Interface 	Opens the MATLAB Interface Analysis Tool. This tool provides an interface between Saber and CosmosScope applications to MATLAB applications.

Mouse Usage

To run CosmosScope on a Unix system, use a three button mouse.

On Windows systems, you may use a two-button mouse, a three-button mouse, or a two-button mouse configured to emulate a three-button mouse.

When using a two-button mouse, access the functionality provided by a third mouse button by holding down the right mouse button and selecting the function from the popup menu that appears.

Mouse Buttons	Function
Left button single click	Select an object
Left button single click with Shift key pressed.	Select multiple objects. Objects in the selected set cannot be un-selected with a left button click
Left button single click with Control key pressed	Select multiple objects Objects in the selected set can be un-selected with a left button click
Left button press and hold and drag	Move an object
Left button double click	1. Open files and directories 2. Bring up Attributes form
Middle mouse button single click	Paste an object
Middle mouse button press and hold and drag	Pan across a window
Right mouse button press and hold	Bring up pop-up menu

Alternatives for Two-Button Mouse Users

The following table provides a description of alternative ways for Windows NT two-button mouse users to access middle mouse button functionality.

Description	Alternative
The middle mouse button is used to paste into a specific location on a graph.	This can also be done with the **Paste** menu item in the **Graph** popup.
Panning the graph surface can be done by the middle mouse button.	This can also be done with the arrow keys. The x axis can be panned with the Range display (x axis scroll bar) if it is turned on with the **Graph > Selected Axes >Sliders > Show** menu item.
The middle mouse button can be used to paste text into the Command Line window.	This can also be done by putting the object on the clipboard and using the **Edit->Paste** menu pull down menu item.
The middle mouse button can be used to paste values and waveforms into the Calculator.	This can also be done with the **Edit > Cut/Copy/Paste** calculator pull down menu entry.
The middle mouse paste is used in selecting items from the Calculator stack and pasting them into the x-register.	This can also be done with the **Edit > Cut/Copy/Paste** pulldown menu item.

Hot Keys

Some operations have shortcut keystroke equivalents (hot keys). The following table lists these hot keys.

Hot Key	Function
`Ctrl+o`	Open design
`Ctrl-a`	Open new graph window
`Ctrl-Del`	Close current graph window
`Ctrl+1`	Iconify current graph window
`Ctrl+f`	Zoom to fit
`Ctrl+z`	Zoom in
`Ctrl+5`	Zoom out

File Types Supported by CosmosScope

Plotfile	Description
`*.*.ai_pl`	Saber Simulator Plot Files
`*.pl.*, *pl`	Saber Simulator Plot Files, from versions preceeding release 5.0
`*.tr*`	HSPICE Transient simulation output files
`*.ac*`	HSPICE AC simulation output files
`*.ft*`	HSPICE FFT simulation output files
`*.sw*`	HSPICE DC simulation output files
`*.m?(0-9a-z)`	HSPICE measurement files
`*.wdb*, *.xp*`	Star-SimXT Plot Files
`*.dmp, *.dump, *.vcd`	Verilog VCD Files. Note: When a `vcd` file (`*.dmp`) is opened, a temporary file (`*.vwdb`) will be generated by Cosmos for reading the data from `*.dmp`. The temporary file cannot be read by CosmosScope directly.
`*.pwdb`	Polaris Plot Files
`*.ai_graph, graph.def`	Graph Save Files
`*.ol`	Outline Save Files
`*.txt`	Text files
`*.csv`	Excel files
`*.s*p, *.s[0-9][0-9]`	TouchStone files
`*.ai_awd`	SaberHDL output files
`*.fsdb`	NanoSim FSDB, Version 2.3 EPIC and VERILOG formats.
`*.sc*`	HSPICE.lin simulation output files.
`*.hb, *.pn*, *.hr*, *.jt*`	HSPICE RF simulation output files.

Appendix A

External Waveform Database API

The External Waveform Database API provides a way to create a dynamically loadable database access package (`dll`) for reading non-Saber data formats into CosmosScope. It contains the following:

- An include file containing structures and interface routines to AIM used to create a database access package to access non-Saber databases. AIM is a scripting language based on Tcl/Tk.
- Examples and guidelines for creating member routines assigned to the format structure defined in the include file.
- A complete example of a database reader.
- Instructions for compiling and linking a database access package.
- Instructions for loading the database access package on invocation of CosmosScope and for setting preferences.

To use the External Waveform Database API, you will also need a `WF_API` license.

Creating a Database Reader

The following steps must be completed to create a database reader.

- Include the include file `formatApi.h` provided with the External Waveform Database API.
- Define the initialization routine and assign member routines to the format structure in the initialization routine.
- Create member routines.

Define Initialization Routine

When the database reader is loaded, the initialization routine must call the routine `PfFormat_CreateExternalFormat()`. This routine registers the new format.

To define the initialization routine, you must complete the following steps:

1. Assign the format name and file extension.
2. Assign member routine names to the `Pf_Format` structure. Guidelines for creating the member routines are provided in the next section.
3. Call the `PfFormat_CreateExternalFormat` routine.

The `Pf_Format` structure and `PfFormat_CreateExternalFormat` routine are shown below:

```
struct Pf_Format {
  char  *name;
  char  *extension;
  PfFormat_GetFormatAttProc    *getFormatAttProc;
  PfFormat_OpenContainerProc   *openContainerProc;
  PfFormat_CloseContainerProc  *closeContainerProc;
  PfFormat_GetContainerAttProc *getContainerAttProc;
  PfFormat_GetWaveformAttProc  *getWaveformAttProc;
  PfFormat_CreateWaveformProc  *createWaveformProc;
  } Pf_Format;

void PfFormat_CreateExternalFormat _ANSI_ARGS_((
  Tcl_Interp *interp,
   Pf_Format *format  /* pointer to a Pf_Format structure which is */
   ));               /* provided by the database access package */
                       /* initialization routine.*/
```

Create Member Routines

When new formats are created, a new `Pf_Format` instance is created. The format name is added to a hash table internal to the application which associates the name with the address of the `Pf_Format` instance. The `Pf_Format` instance contains the name of the format and the addresses of all the format's member routines that are provided by the database reader. The member routines implement the querying of the format, container, and waveform attributes as well as the creation of waveforms. The member routines are called through the External Waveform Database API when attributes or data are needed by the application.

All member routines return TCL_ERROR or TCL_OK as return values to indicate the status of the call.

You will need to create the member routines described below.

GetFormatAttProc

The GetFormatAttProc is used to get attributes about a given format. Access to this information is provided by the AIM command pf:format. When called, the database reader must provide the corresponding value in the form of a string. You can provide support in the database reader for other attributes like version, author, etc., if you wish to be able to query these using the pf:format command.

The interface for this routine is shown below.

```
typedef int (PfFormat_GetFormatAttProc) (
  Tcl_Interp *interp,      /* pointer to interpreter */
  char *attribute,         /* attribute name string */
  PfFormat_Value *value);  /* value returned by routine */
```

OpenContainerProc

The OpenContainerProc is used to initiate access to an external data file. This routine is called when the AIM command pf:open is performed on an external data file. The full pathname to the external data file is passed to the routine. The database reader uses the pathname to open the file. It then performs operations and obtains data, creates a data structure with data specific to the particular data file, and returns the address of this structure back to the calling routine as the client data. (At this point, the application has not requested any information about the waveforms or the container, so there may be no need to read any data from the data file at this time). When called, the address to this structure is passed to the format's other member routines. This way, the member routines can uniquely identify which container is being referenced as well as pass container specific data between themselves.

The interface for this routine is shown below.

```
typedef int (PfFormat_OpenContainerProc) (
  Tcl_Interp *interp,     /* pointer to interpreter */
  char *pathname,         /* path name to data file */
  ClientData *clientdata); /* address of data structure specific to */
                          /* the opened container returned by the */
                          /* Proc. */
```

CloseContainerProc

The `CloseContainerProc` is called when the container is to be closed using the AIM command `pf:close`. It allows the database reader to de-allocate anything allocated for its own use.

The interface for this routine is shown below.

```
typedef int (PfFormat_CloseContainerProc) (
  Tcl_Interp *interp,      /* pointer to interpreter */
  ClientData clientdata);  /* address of data structure specific */
                           /* to the opened container */
                           /* returned by the Proc. */
```

GetContainerAttProc

The `GetContainerAttProc` is used to get attributes about a specific container. This routine will be called when the AIM command `pf:info` is performed on a container associated with an external format. The set of required attributes the database reader must provide are `-created`, `-design`, `-filenames`, `-nwaveforms`, `-pfname`, and `-waveforms`.

The interface for this routine is shown below.

```
typedef int (PfFormat_GetContainerAttProc) (
  Tcl_Interp *interp,     /* pointer to interpreter */
  ClientData clientdata,  /* address of data specific to the */
                          /*container. */
  char *attribute,        /* attribute name string */
  PfFormat_Value *value); /* value returned by routine */
```

GetWaveformAttProc

The `GetWaveformAttProc` is used to get attributes about a specific waveform. This routine is called when the AIM command `wf:info` is performed on a waveform associated with an external format. The set of required attributes that the database reader must provide are `-datatype`, `-got_xaxis`, `-nsegments`, `-wftype`, `-xname`, `-xunit`, `-xscale`, `-yname`, `-yunit`, and `-yscale`.

The interface for this routine is shown below.

```
typedef int (WfX_GetWaveformAttProc) (
  Tcl_Interp *interp,    /* pointer to interpreter */
  ClientData clientdata, /* address of data specific to the container.*/
  char *wfname,          /* waveform name string */
  char *attribute,       /* attribute name string */
  PfFormat_Value *value);/* value returned by routine */
```

CreateWaveformProc

The `CreateWaveformProc` is called when a complete waveform is to be to be loaded into memory (for example, when a waveform is plotted, or used in a calculation, or used in a measurement). To create the waveform, use the set of waveform creation routines described in the next section in this routine.

The interface for this routine is shown below.

```
typedef int (PfFormat_CreateWaveformProc) (
  Tcl_Interp *interp,   /* pointer to interpreter */
  ClientData clientdata,/* address of data specific to the container.*/
  char *wfname,         /* waveform name string */
  Wf_rec *wfptr));      /* waveform record returned by the database */
                        /* reader*/
```

Waveform Creation Routines

Interfaces to the waveform creation routines are provided in the include file `formatApi.h`. You can call these routines as needed from the database reader member routine `CreateWaveformProc` to construct the waveform object. (This approach to creating waveforms is similar to using the AIM `wf` command. This command is documented in the online documentation provided with CosmosScope).

Examples of constructing a simple, non-parameterized, analog waveform and a parameterized (family of curves) waveform are shown below.

Non-parameterized Waveform Routine

Shown below is an example of a code fragment from a waveform creation routine illustrating how a simple, non-parameterized, analog waveform is created.

```
/* For a simple, non-parameterized, analog waveform */
{
  ...

  int      i;

  Wf_rec   wf;        /* waveform record returned by Wf_Create() */
  wf_type  stype;     /* Type of waveform and data */
  char     xname[24]; /* name for x axis */
  char     xunit[24]; /* name for x units */
  char     yname[128];/* name for y axis */
  char     yunit[24]; /* name for y units */
```

```
  int      numpts;    /* number of points to add to waveform */
  PfFormat_Value  xval;  /* x values for waveform */
  PfFormat_Value  yval;  /* y values for waveform */

  ...

  stype = WF_ANALOG_REAL;


  ...

  /* Get name and units for x and y data */
  strcpy(xname,...
  strcpy(yname,...
  strcpy(xunits,...
  strcpy(yunits,...
  ...

  wf = WfX_Create(interp,stype,xname,xunit,WF_LINEAR,yname,yunit,1);

  xval.type = WFX_DOUBLE;
  yval.type = WFX_DOUBLE;
  for (i=0; i<numpts; i++) {


       /* Get data for xval and yval and assign */
       ...
       xval.u.doubleValue = ...
       yval.u.doubleValue = ...

       WfX_AddValue(interp, wf, &xval, &yval);
  }

}
```

Interfaces to the routines provided to create and add points to simple, non-parameterized waveforms are shown below.

WfX_Create()

The routine `WfX_Create()` creates and initializes a waveform with the specified arguments. The waveform is not complete, however, until the

`CreateWaveformProc` returns. The `Wf_rec` structure is passed to the other waveform creation routines as the waveform is constructed.

```
EXTERN Wf_rec WfX_Create(   /* Initiates the creation of a waveform.*/
  Tcl_Interp *interp,       /* address of interpreter. */
  wf_type stype,            /* waveform type, ex: WF_ANALOG_REAL */
  char *x_name,             /* x axis name, example: "t". */
  char *x_units,            /* x axis units, example: "s". */
  wf_data_scale xtag,       /* x axis scaling, WF_LINEAR/WF_LOG. */
  char *y_name,             /* y axis name, example: "output". */
  char *y_units,            /* y axis units, example: "V". */
  int width);               /* width of y values, 1 for all types */
                            /* except WF_BUS, WF_REG, WF_REGINT */
```

Wf_CreateDgt()

The routine Wf_CreateDgt() does the similar thing with routine Wf_Create(), except it can create a digital/bus waveform with particular logical type that is specified by the last argument.

```
EXTERN Wf_rec Wf_Create(/* Initiates the creation of a waveform.  */
Tcl_Interp *interp,   /* address of interpreter. */
wf_type stype,           /* waveform type, ex: WF_ANALOG_REAL */
char *x_name,            /* x axis name, example: "t".*/
char *x_units,           /* x axis units, example: "s".*/
wf_data_scale xtag,      /* x axis scaling, WF_LINEAR/WF_LOG.*/
char *y_name,            /* y axis name, example: "output".*/
char *y_units,           /* y axis units, example: "V".*/
int width,               /* width of y values, 1 for all types except*/
                         /* WF_BUS, WF_REG, WF_REGINT */
char *datatype);         /* Logical type for digital/bus signals. */
                             /* example: “logic_4”, “std_logic”, /
                                   “nanosim_logic_12” */
```

WfX_AddValue()

The routine `WfX_AddValue()` adds a data point to the waveform. `xval` and `yval` contain values in whatever type is appropriate for the waveform type. For waveform type WF_BUS the value is a binary number with the number of characters equal to the width of the bus. For example, a valid value for a WF_BUS waveform with a width of 8 is “10011011”. For waveform types WF_REG, and WF_REGINT, the value is given in hex characters. For

example, a valid value for a WF_REG or WF_REGINT waveform with a width of 8 is “AA”.

```
 EXTERN int WfX_AddValue(   /* adds a data point to the wf_data
waveform*/
   Tcl_Interp *interp,      /* address for the interpreter */
   wfrec *wf_data,          /* handle for the waveform */
   PfFormat_Value *xval,    /* x value for the data point */
   PfFormat_Value *yval);   /* y value for the data point */
```

Wf_AddValues()

The routine Wf_AddValues() adds a bunch of data points to the waveform at one time. The x and y values of the data points are passed in by the void pointers “xval” and “yval” arguments. Usually, the “xval” is a double type pointer. The “yval” is a double type pointer for real type waveforms, or an Aim_Complex type pointer for complex type waveforms, or a “char **” type pointer for digital/bus/register type waveforms. The number of points inserted is passed in by the argument “valueCount”. The void pointers “xval” and “yval” need to be freed after used. Calling Wf_AddValues() routine would have much better performance than calling wf_AddValue().

```
EXTERN int Wf_AddValues(/* adds a data point to the wf_data waveform */
Tcl_Interp *interp,     /* address for the interpreter              */
wfrec *wf_data,             /* handle for the waveform              */
void *xval,                 /* x values for the data points         */
void *yval,                 /* y values for the data points         */
int valueCount);            /* number of points contained in the    /
                                       xval and yval                 */
```

When using Wf_AddValues() instead of Wf_AddValue(), the last loop in the previous example code would be changed to:

```
double *new_xval=NULL;  /*declared in the position where xval and yval /
                            are declared*/
double *new_yval=NULL;
for (i=0; i<numpts; i++) {
      new_xval = (double )malloc(numpts*sizeof(double));
      new_yval = (double )malloc(numpts*sizeof(double));

      /* Get data for new_xval[i] and new_yval[i] and assign */
      new_xval[i] = …
      new_yval[i]= …
}
WfX_AddValues(interp, wf, (void*)new_xval, (void*)new_yval, numpts);
if (new_xval != NULL)
      free(new_xval);
if (new_yval != NULL)
      free(new_yval);
```

Parameterized Waveform Routine

The parameter routines are used to create parameterized waveforms (a family of curves). After a parameterized waveform is initialized using the `WfX_Create()` routine, the `WfX_AddNumberParameter()`, `WfX_AddSetParameter()`, `WfX_AddStringParameter()` routines are used to define the set of parameter values for the waveform. The `WfX_NextParameterValue()` routine is used to step through each parameter value after adding the waveform data points.

Shown below is an example of a code fragment from a waveform creation routine illustrating how a parameterized analog waveform is created.

```
/* For parameterized, analog waveform */
{

  ...

  Wf_rec          wf;          /* Waveform handle returned from */
                               /* WfX_Create()*/
  wf_type         stype;       /* Type of waveform and data */
  char            xname[24];   /* name for x axis */
  char            xunit[24];   /* name for x units */
  char            yname[128];  /* name for y axis */
  char            yunit[24];   /* name for y units */
  int             numpts;      /* number of points to add to waveform */
  PfFormat_Value  xval;        /* x value for waveform */
  PfFormat_Value  yval;        /* y value for waveform */
  char            parName[24]; /* name for parameter value */
  char            parUnit[24]; /* units for parameter value */
  int             parValNum;   /* number of parameter values */
  double          *parValues;  /* parameter values */

  ...

  stype = WF_ANALOG_REAL;

  ...

  /* Get name and units for x and y data */
  strcpy(xname,...
  strcpy(yname,...
  strcpy(xunits,...
  strcpy(yunits,...

  ...

  /* Get name, units, number of parameters and values */
  strcpy(parName,...
  strcpy(parUnit,...
  parValNum = ...
  for (i=0;i<ValNum;i++) {
  parValues[i] = ...
  }
```

```
    ...

    /* Initialize waveform */
    wf = WfX_Create(interp,stype,xname,xunit,WF_LINEAR,yname,yunit,1);


   /* Specify parameters associated with waveform */
   WfX_AddNumberParameter(interp,wf,parName,parUnit,parValNum,
parValues);

   for (j=0; j<parValNum; j++) {
        for (i=0; i<numpts; i++) {

             /* Get data for xval and yval and assign */
             ...

             WfX_AddValue(interp, wf, &xval, &yval);
        }
        if (j < parValNum - 1) {
             WfX_NextParameterValue (interp, wf);
        }
   }
}
```

Interfaces to the routines provided to create and add points to parameterized waveforms are shown below.

WfX_AddNumberParameter()

The `WfX_AddNumberParameter()` routine is used to define a set of numeric parameter values for a waveform that are continuous.

```
EXTERN int WfX_AddNumberParameter(
  Tcl_Interp *interp,
  wfrec *wf_data,
  char *par_name,
  char *par_unit,
  int par_argc,
  double *par_argv);
```

WfX_AddSetParameter()

The `WfX_AddSetParameter()` routine is used to define a set of numeric parameter values for a waveform that are discrete.

```
EXTERN int WfX_AddSetParameter(
  Tcl_Interp *interp,
  wfrec *wf_data,
  char *par_name,
  char *par_unit,
  int par_argc,
  double *par_argv);
```

WfX_AddStringParameter()

The `WfX_AddStringParameter()` routine is used to define a set of string parameter values for a waveform that are discrete.

```
EXTERN int WfX_AddStringParameter(
  Tcl_Interp *interp,
  wfrec *wf_data,
  char *par_name,
  int par_argc,
  char **par_argv);
```

WfX_NextParameterValue()

The routine `WfX_NextParameterValue()` steps the waveform record to the next parameter value.

```
EXTERN int WfX_NextParameterValue(
  Tcl_Interp *interp,
  wfrec *wf_data);
```

Compiling and Linking the Database Access Package (dll)

To compile a Database Access Package (`dll`) written in the C programming language *on an NT system*, complete the following steps:

1. Include a header in the routine exactly as shown below where *package* is the name of your routine:

```
__declspec(dllexport) int package_Init (Tcl_Interp *interp)
{
}
```

2. Execute the following commands:

```
cl /MD /D "WIN32" /c "package.c"
link %SABER_HOME%\lib\libai_tcl.lib
   %SABER_HOME%\lib\libai_aimtcl.lib /dll "package.obj"
```

On a Solaris system, to compile a Database Access Package (`dll`) written in the C programming language, complete the following steps:

1. Compile the source (in this example, `fmt1.c`) by executing the following:

```
cc -K PIC -I $SABER_HOME/include -c fmtl.c -o fmt1.o
```

2. Create a shareable library by executing the following:

```
ld -G -o fmtl.so fmt1.o
```

3. To interactively load the package, invoke CosmosScope and use the Command Line tool to enter the following:

```
aim:load fmtl -exact
```

   See the next section "Loading the Database Access Package" for a method to load the package when CosmosScope is invoked.

On a Hewlett Packard system, to compile a Database Access Package (`dll`) written in the C programming language, complete the following steps:

1. Compile the source (in this example, `fmt1.c`) by executing the following:

```
cc -Aa +Z -I $SABER_HOME/include -c fmtl.c
```

2. Create a shareable library by executing the following:

```
ld -b -o fmtl.sl fmt1.o
```

3. To interactively load the package, invoke CosmosScope and use the Command Line tool to enter the following:

```
aim:load fmtl -exact
```

See the next section “Loading the Database Access Package” for a method to load the package when CosmosScope is invoked.

Loading the Database Access Package

To load the database access package when CosmosScope is invoked, complete the following steps.

1. Add the `aim:load` command to a CosmosScope startup file. The `aimload` command syntax is shown below.

`aim:load` *package (initRoutine)* `-exact`

The required entry *package* is the name of the database access package (`dll`) without an extension. The optional entry *initRoutine* is the name of the database reader routine. The entry *initRoutine* is not required if this name corresponds to the name of the `dll` as *package*`_Init`. The `-exact` option (required) specifies that the package name must be used exactly as specified in the `aimload` command.

You can add the command to either of the following CosmosScope startup files:

- Your site startup file `scopeRc.site` located in a site-accessible directory included in the `PATH` environment variable on your users’ systems.
- The user’s startup file `.scopeRc_user` located in the user’s home directory.

2. Modify the CosmosScope startup file as needed to specify the default filter in the waveform browser in CosmosScope. The default will be one of the following:

- If no external formats have been loaded, the default format will be “Plotfiles” (the Saber format).

- If you have loaded one or more external formats and have not specified a default format, the default format will be the last format loaded.
- If you have specified a default format as described below, the specified format will be the default format.

To specify a default format, include the following command in the CosmosScope startup file in the line after the `aimload` command (see Step 1 above):

`set Scope(PfFilter)` *format_name*

If you want the default format to be the Saber format, replace *format_name* with `Plotfiles`. If you want to specify an external filter, use the name assigned to the format in the `Pf_Format` structure (see "Define Initialization Routine" on page 2)

Once these steps have been completed, the `aimload` command will be executed each time CosmosScope is invoked and the initialization routine will be run. The new format will be registered and a new instance of the format object with appropriate attributes will be created. This instance will be added to any others created previously.

Files Provided with the Saber Software

The following file contains the `include` file provide with the External Waveform Database API package:

```
$SABER_HOME/include/formatApi.h
```

The following two files contain an example database reader and a data file showing the datafile format used by the database reader:

```
$SABER_HOME/example/SaberScope/DatabaseAPI/fmt1.c
$SABER_HOME/example/SaberScope/DatabaseAPI/fmt1.dat
```

Appendix B

ASCII File Export and Import

Export

CosmosScope allows you to export selected waveforms and plotfiles into a text file to support comparison of your simulation results to lab measurements. The **Edit->Scope Preferences** menu enables you to specify the data precision and separator settings.

Set Export Preferences

Use the **Edit->Scope Preferences** menu, under the Reader tab, to set the Text Writer/Reader options:

- Set your preferred Writing Precision value; the default precision is **6**.
- Select the Name/Unit Separator value from the available choices. The default is `.

Exporting Waveforms

To export selected waveforms from the Graph Window, a Signal List, or the Calculator, use the **File > Save > Plotfile(*.txt)** menu; specify the path and file name. CosmosScope will save your selected waveforms to an ASCII file with the extension `".txt."`

Note the following conditions or requirements:

- For multiple analog signal selections, the waveforms must have the same x-axis names.
- Digital signal(s) in selected waveforms are saved without an x-axis specification; CosmosScope will open this saved text file with a default x-axis t(s).
- The ASCII plotfile format does not support multi-member waveforms. Each waveform in an ASCII file must contain only one segment. If a

waveform from a different plotfile format contains more than one segment, each segment will be written as a separate waveform when it is exported to an ASCII format.

The syntax for the AIM command alternative is:

```
CSV:Write file wfHandle1 wfHandle2 wfHandle3 …
```

where `file` specifies the path name of the ASCII file. At least one waveform handle is required.

For example:

```
CSV:Write [path]/single_amp.txt WF:1:1 WF:1:2 WF:1:3
```

where `[path]` specifies the location of the ASCII file, `single_amp.txt`.

Exporting Plotfiles

To export a plotfile into an ASCII file format, within the Signal Manager window, select the desired plotfile. Choose the Signal Manager menu option **File > Save As**; then, select file type "**Text (*.txt)**" and specify a path and file name.

***NOTE**: You can select only one plotfile at a time.*

An AIM command could be used directly by specifying the plotfile handle and file name path, in the format:

```
ScopeSigMgr:SaveAsText pfHandle path
```

For example:

```
ScopeSigMgr:SaveAsText PF:1 /home/usr/single_amp.txt
```

Import

Scope allows import of an ASCII file with an extension of `.txt` or `.csv`.

To import your file, select the CosmosScope **File->Open->Plotfiles.** Set the Files of Type setting to **Text/Excel (*.txt, *.csv)**. Navigate/browse to the desired file.

The ASCII file you specify may have been generated from CosmosScope (via the export process described above) or one you have generated via another means, provided it follows the required format described below. The imported

file must include two parts: the waveform Descriptor/Header, and the waveform data.

Waveform Descriptor / Header:

The header contains each waveform's x-axis name and unit, y-axis name and unit, the waveform type, and the data type.

The header is a string of (nwfs + 1) elements, where nwfs provides the total number of waveforms (Dependent Variables), plus one Independent variable element.

Each header element sub-string is delineated by a separator. The five options for separators are "`", "'", ":", ";", and "|". Only one separator can be used within a given file. This separator must match your **Scope Preferences** setting, described above.

Independent variable element:

The first element in the header specifies the independent variables, including x name and unit, separated by the separator you've specified. For example:

```
xname`xunit
```

Either or both the xname and xunit can be absent. If only the xname is missing, CosmosScope will assign "t" by default. However, if both xname and xunit are missing, the separator must nonetheless be included to indicate presence of this independent variable element.

Dependent variable element:

The dependent variable element includes:

- the waveform name (required)
- type (optional: options are analog, digital, or bus; default is analog)
- unit (optional)
- data type (optional: options are real or complex; default is real)

For example:

```
wfname1`wftype1`yunit1`datatype1
```

For the digital or bus signal, instead of waveform unit and data type, the logic type information should be provided in the element, e.g.

```
wfname2`bus`logic_4.
```

CosmosScope supports 3 logic types:

- logic_4
- nanosim_logic_12
- std_logic

If the logic type is not specified, CosmosScope will assign a default of logic_4. Below is an example of a complete header:

```
t‘s i(out)‘analog‘A‘real d1‘digital‘logic_4
```

Data

The Data section of the file contains columns of values. The first column reflects the value of the independent variable. The remaining columns contain the dependent value(s) for each signal, in the same order as they appear in the header session. “NULL” is used in the column when there is no dependent value for the respective independent value for the signal.

If the data type of a waveform is complex, where you may use different conventions to express that complex data, CosmosScope provides several choices. The following list provides examples of the possible formats. (Note, no space is allowed before or after the “+” or “-“):

2+3j or 2-3j

2+j+3 or 2+j-3

2+j3 or 2-j3

2

j

3j or -3j

j3 or j-3

Sample ASCII Import File

A sample ascii import file format is shown below; the three elements on each line are separated by either a space or tab:

t`si (v_clock.v_clock)`A n_1`V

0.004 -0.012 12

```
0.004 0 0
0.00400001 0 0
0.00400011 0 0
0.0045 0 0
0.0045 -0.012 12
0.00450001 -0.012 12
0.00450011 -0.012 12
```

Index

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z

BOOKSHELF

CosmosScope™ AIM Reference Manual

CosmosScope™ Calculator Reference Manual

CosmosScope™ MATLAB® Interface User Manual

CosmosScope™ Reference Manual

Saber® and CosmosScope™ AIM User Guide

Saber® and CosmosScope™ Command Line Tool User Guide

Saber® and CosmosScope™ Draw Tool User Guide

Saber® and CosmosScope™ Macro Recorder User Guide

Saber® and CosmosScope™ Report Tool Reference Manual

Saber® and CosmosScope™ StateAMS Reference Manual