CosmosScope™ User Guide

Version F-2011.09, September 2011

SYNOPSYS®

Copyright Notice and Proprietary Information

Copyright © 2011 Synopsys, Inc. All rights reserved. This software and documentation contain confidential and proprietary information that is the property of Synopsys, Inc. The software and documentation are furnished under a license agreement and may be used or copied only in accordance with the terms of the license agreement. No part of the software and documentation may be reproduced, transmitted, or translated, in any form or by any means, electronic, mechanical, manual, optical, or otherwise, without prior written permission of Synopsys, Inc., or as expressly provided by the license agreement.

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, AEON, AMPS, Astro, Behavior Extracting Synthesis Technology, Cadabra, CATS, Certify, CHIPit, CoMET, Confirma, CODE V, Design Compiler, DesignSphere, DesignWare, Eclypse, EMBED-IT!, Formality, Galaxy Custom Designer, Global Synthesis, HAPS, HapsTrak, HDL Analyst, HSIM, HSPICE, Identify, Leda, LightTools, MAST, METeor, ModelTools, NanoSim, NOVeA, OpenVera, ORA, PathMill, Physical Compiler, PrimeTime, SCOPE, Simply Better Results, SiVL, SNUG, SolvNet, Sonic Focus, STAR Memory System, Syndicated, Synplicity, the Synplicity logo, Synplify, Synplify Pro, Synthesis Constraints Optimization Environment, TetraMAX, UMRBus, VCS, Vera, and YIELDirector are registered trademarks of Synopsys, Inc.

Trademarks (™)

AFGen, Apollo, ARC, ASAP, Astro-Rail, Astro-Xtalk, Aurora, AvanWaves, BEST, Columbia, Columbia-CE, Cosmos, CosmosLE, CosmosScope, CRITIC, CustomExplorer, CustomSim, DC Expert, DC Professional, DC Ultra, Design Analyzer, Design Vision, DesignerHDL, DesignPower, DFTMAX, Direct Silicon Access, Discovery, Encore, EPIC, Galaxy, HANEX, HDL Compiler, Hercules, Hierarchical Optimization Technology, High-performance ASIC Prototyping System, HSIM^{plus}, i-Virtual Stepper, IICE, in-Sync, iN-Tandem, Intelli, Jupiter, Jupiter-DP, JupiterXT, JupiterXT-ASIC, Liberty, Libra-Passport, Library Compiler, Macro-PLUS, Magellan, Mars, Mars-Rail, Mars-Xtalk, Milkyway, ModelSource, Module Compiler, MultiPoint, ORAengineering, Physical Analyst, Planet, Planet-PL, Polaris, Power Compiler, Raphael, RippledMixer, Saturn, Scirocco, Scirocco-i, SiWare, Star-RCXT, Star-SimXT, StarRC, System Compiler, System Designer, Taurus, TotalRecall, TSUPREM-4, VCSi, VHDL Compiler, VMC, and Worksheet Buffer 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.
Saber is a registered trademark of SabreMark Limited Partnership and is used under license.
All other product or company names may be trademarks of their respective owners.

Contents

1. Getting Started with CosmosScope 1

Invoking CosmosScope 1

Windows Platforms 1

UNIX Platforms 2

Setting Preferences for Multiple Users. 3

Connecting Third-Party Tools to CosmosScope 3

Setting Up a Connection 3

Testing the Connection. 4

Opening a Plot File 5

2. Tutorials 7

Taking Measurements in CosmosScope 7

Performing a Pareto Analysis. 8

Simulating in HSPICE 8

Analyzing in CosmosScope 9

Viewing Saber Simulator Results. 11

Setting up the Saber Simulator Data 11

Viewing Saber Transient Analysis Waveforms 12

Viewing Saber AC Analysis Waveforms. 13

Performing Measurements on a Waveform 14

Viewing HSPICE Results. 17

Setting up the Design Data 17

Viewing HSPICE Transient Analysis Waveforms 17

Viewing AC Analysis Waveforms 18

Performing Measurements on an HSPICE Waveform 20

3. Signals 23

Converting Analog Signals to Digital 24

Converting Digital Signals to Analog 24

Converting Single-Bit Digital Signals to Analog 24

Converting Digital Bus Signals to Analog 25
Converting Frequency Domain Signals to the Steady-State Time Domain . . . 25
Changing Signal Appearance 25
Changing Signal Zoom Level 26
Panning Signals 26
Changing Signal Display Colors 27
Defining Custom Signal Display Colors 27
Defining Analog Signal Colors 27
Defining Digital Signal Colors 28
Moving Signals to Graph Regions 28
Specifying Analog Signal Fill Patterns 29
Combining Digital Signals into a Bus 29
Displaying the Digital Signal Grid 29
Changing Symbol Styles 30
Changing Signal Colors 30

4. Graphs 31
Displaying Graphs 31
Saving Graphs 32
Opening Outlines 32
Saving Outlines 34
Placing Trace Markers 34
Changing Trace Height Display 35
Annotating Graphs 35
Exporting Images 35
Changing Graph Font Face and Style 35
Changing Graph Zoom Level 35
Displaying the Slider Bar 36

5. Waveforms 37
Comparing Waveforms 37

Creating Multi-Member Parameter Files 39
Merging All or Selected Plotfile Waveforms into Multi-Member Waveforms . . 39
Creating Multi-Member Waveforms 44

Index 47

Contents

1

Getting Started with CosmosScope

Explains how to invoke CosmosScope, set user preferences, connect to third-party tools and open a plot file.

This section covers the following topics:

- Invoking CosmosScope
- Setting Preferences for Multiple Users
- Connecting Third-Party Tools to CosmosScope
- Opening a Plot File

Invoking CosmosScope

Choose one of the following platforms for instructions on how to invoke CosmosScope:

- Windows Platforms
- UNIX Platforms

Windows Platforms

On a Windows system, use the UNIX Platforms command-line invocation or choose:

Programs > {install_location} > Scope

or

Programs > {install_location} > CosmosScope

UNIX Platforms

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] or [-script < aimfile] [-app application
name] [-nosplash]
```

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 CosmosScope window.
-script aimfile or -script < aimfile	Executes the specified AIM script on start-up.
-nosplash	Disables the splash screen on startup.
-app application name	Specifies the application that CosmosScope is integrated with. The value can be saber, cosmos, saberhdl, or custom.

Setting Preferences for Multiple Users

You can set the CosmosScope preferences for multiple users at a single site using the AI_SITE_PATH variable. See the *AIM Reference* for more information.

Connecting Third-Party Tools to CosmosScope

CosmosScope provides a communication protocol using AIM (an extension of TCL/TK), which allows another application to invoke CosmosScope, send commands, get results back from CosmosScope, and monitor job status. CosmosScope uses a package called AimAppCom for communications.

This section covers the following topics:

- Setting Up a Connection
- Testing the Connection

Setting Up a Connection

Note: Examples in this section refer to setting up a connection between CosmosScope and a Cosmos installation.

To set up a connection from a third-party tool to CosmosScope:

1. Ensure your TCL environment is set up properly.
2. Invoke the following applications:
   - CosmosScope

     For example:

     ```
     > cscope
     ```

   - TCL shell (version 8.4 or later)

     For example:

     ```
     > tclsh
     ```
3. In the TCL shell, load a shared library.

For example:

```
% load <cosmos_install>/lib/IA.32/libdp.so
```

4. In the TCL shell, source the following interfaces:
   - distributed process utility

     For example:

     ```
     % source <cosmos install>/etc/tcl/dp.tcl
     ```

   - AimAppCom communications package

     For example:

     ```
     % source <cosmos install>/etc/tcl/AimAppCom.tcl
     ```

   - AIM shell interface

     For example:

     ```
     % source <cosmos install>/etc/tcl/AimAppComExt.tcl
     ```

   The connection between your third-party tool and CosmosScope is complete. Continue to Testing the Connection to see examples of how you can use this connection.

Testing the Connection

From within graphic TCL, try entering one or more of the following command lines to ensure the connection is complete:

- Search for a running CosmosScope server and return its handle:

  ```
  % lindex [AimAppCom:FindServer waveformGrapher] 0
  ```

  The result is `% s0` (or similar).

- Place a graph in CosmosScope:

  ```
  % AimAppCom:SendServer s0 Scope:NewGraph
  ```

  A new graph window opens in CosmosScope.

- Open a plotfile in CosmosScope:

  ```
  % AimAppCom:SendServer s0 ScopeSigMgr:loadpffile "path /
  inv.tr0"0
  ```

  A plotfile opens in CosmosScope.

Opening a Plot File

To open a plot file:

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

2

Tutorials

Provides various tutorials on how to take measurements, perform pareto analysis, and view Saber and HSPICE simulation results using CosmosScope.

This section covers the following topics:

- Taking Measurements in CosmosScope
- Performing a Pareto Analysis
- Viewing Saber Simulator Results
- Viewing HSPICE Results

Taking Measurements in CosmosScope

This tutorial explains how to use CosmosScope to take measurements.

To take a measurement in CosmosScope:

1. Choose **File > Open > Plotfiles...** to open a plotfile that contains the signal(s) you want to measure. The Signal Manager opens, which lists all the signal data that it contains.
2. Plot the signal.

   You can double-click a signal, or select a signal name and click the **Plot** button located at the bottom of the Signal Manager.
3. Click the Measurement Tool icon at the bottom of the main CosmosScope window to open the Measurement Tool.
4. In the Measurement Tool dialog box, click the Measurement button, and choose one of the available measurements. Some measurements might not be available based on the kind of signal you are measuring.

5. In the Active graph text box, specify the graph that contains the signal you want to measure. Specify the signal you want to measure in the Signal text boxes.
6. Change any needed parameters in the Measurement Tool dialog box to produce the desired measurement result.

   See the *CosmosScope Tools Reference* for information on the parameters used for each measurement.
7. Click **Apply** to make the measurement.

Performing a Pareto Analysis

This tutorial explains how to run a Pareto analysis on a Monte Carlo simulation using an OpAmp and consists of the following sections:

Before You Begin

The following items are required for this tutorial:

- CosmosScope version 2007.03 or later
- HSPICE version 2007.03 or later
- The opampmc.sp file, which is located in the following HSPICE demo directory:

  `<installation>/hspice/demo/hspice/variability`

This section covers the following topics:

- Simulating in HSPICE
- Analyzing in CosmosScope

Simulating in HSPICE

Before peforming a Pareto measurement and analysis in CosmosScope, you first need to produce the plotfile and parameter file.

To produce the plotfile and parameter file:

1. Start HSPICE.
2. Enter the following line at the command-line prompt:

   ```
   hspice -i opampmc.sp
   ```

The opampmc.ms0 (plotfile) and Opampmc.mcs0 (parameter file) are produced.

3. Continue to the next section, Analyzing in CosmosScope.

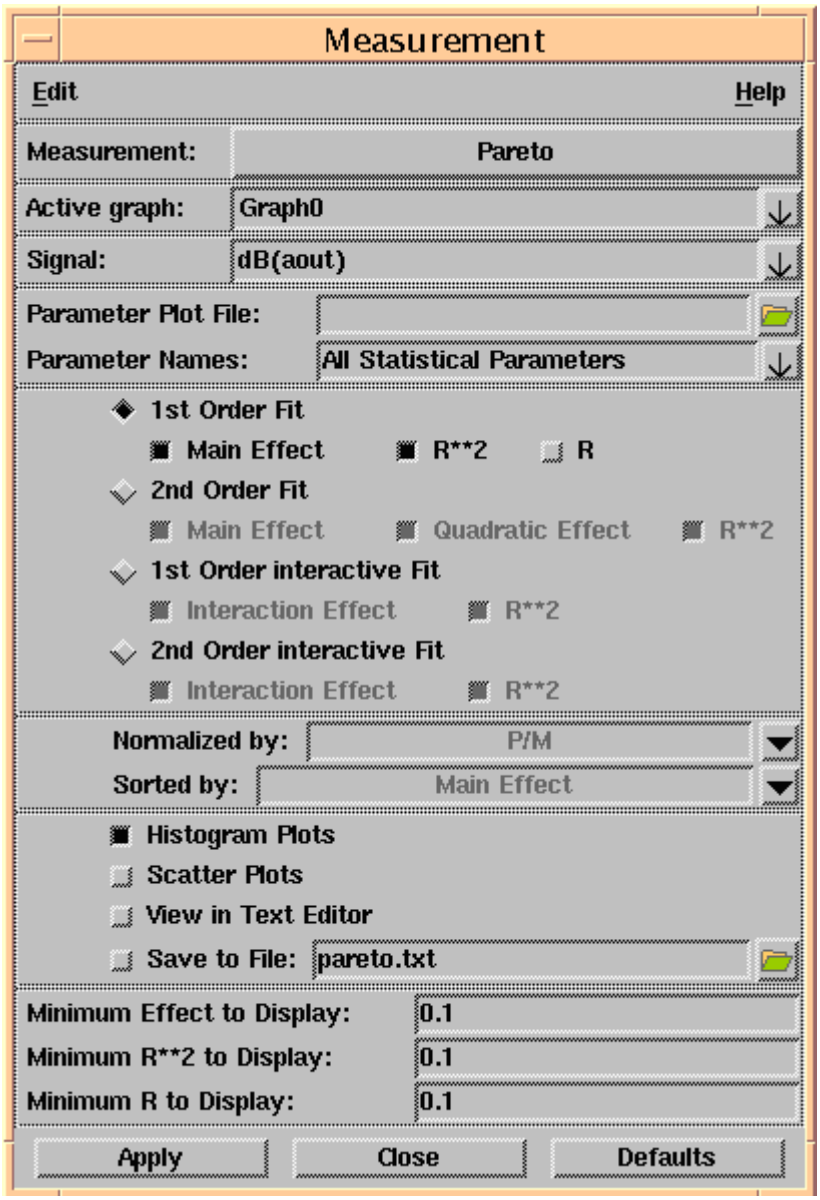
Analyzing in CosmosScope

To perform a Pareto analysis in CosmosScope:

1. Start CosmosScope.
2. Choose **File > Open > Plotfiles...** to open the opampmc.ms0 file, which is an HSPICE meas file. The Signal Manager lists the seven signals that opampmc.ms0 contains.
3. Plot the systoffset2 signal.

   You can double-click a signal, or select a signal name and click the **Plot** button located at the bottom of the Signal Manager.
4. Click the Measurement Tool icon button at the bottom of the main CosmosScope window to open the Measurement Tool.
5. In the Measurement Tool dialog box, click the Measurement button, and choose **Statistics > Pareto**. The Measurement Tool dialog box shows the Pareto measurement options.
6. Ensure Graph0 is specified in the Active graph text box and sysoffset2 is specified in the Signal graph text box.
7. Click the folder icon to the right of the Parameter Plot File text field. Browse to and select the Opampmc.mcs0 parameter file.
8. Click the down arrow to the right of the Parameter Names text field to display a list of all the parameters that Opampmc.mcs0 contains.
9. Select all the parameters in the Select List dialog box, and click **OK**.

10. Change the remaining options to match those in the following figure. Ensure zero (0) is specified for in Minimum R to Display text field.



11. Click **Apply** to make the measurement.

    A histogram of the measurement appears in the CosmosScope main window.

12. Take a look at the results.

Zoom the histogram to fit to get the full display and see what parameters have the most effect on system offset. You can also open the pareto.txt file and see the numerical results (located in column R).

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 section covers 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 the `aout` signal.
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:
   - 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.
   - Click and hold the right mouse button to open the **Signal** menu.
   - Select the Delete Signal item.
5. Do the following to see how you can plot additional waveforms to the Graph window:
   - 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.
   - Delete the in and aout waveforms when you have finished viewing them.
6. Look at the aout (dBV):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

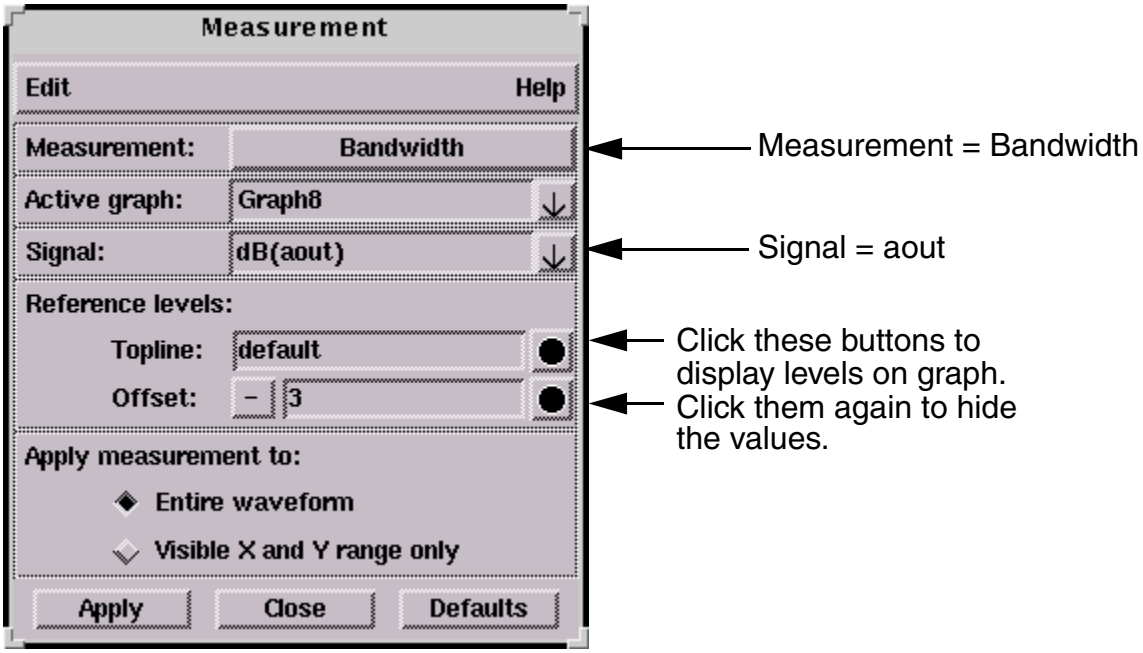
The Measurement Tool allows you to perform various measurements on a waveform. Check the bandwidth and gain of the single-stage amplifier output signal (aout) as follows:

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. From the Measurement Tool window, choose **Measurement > Frequency Domain > Gain Margin**.

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

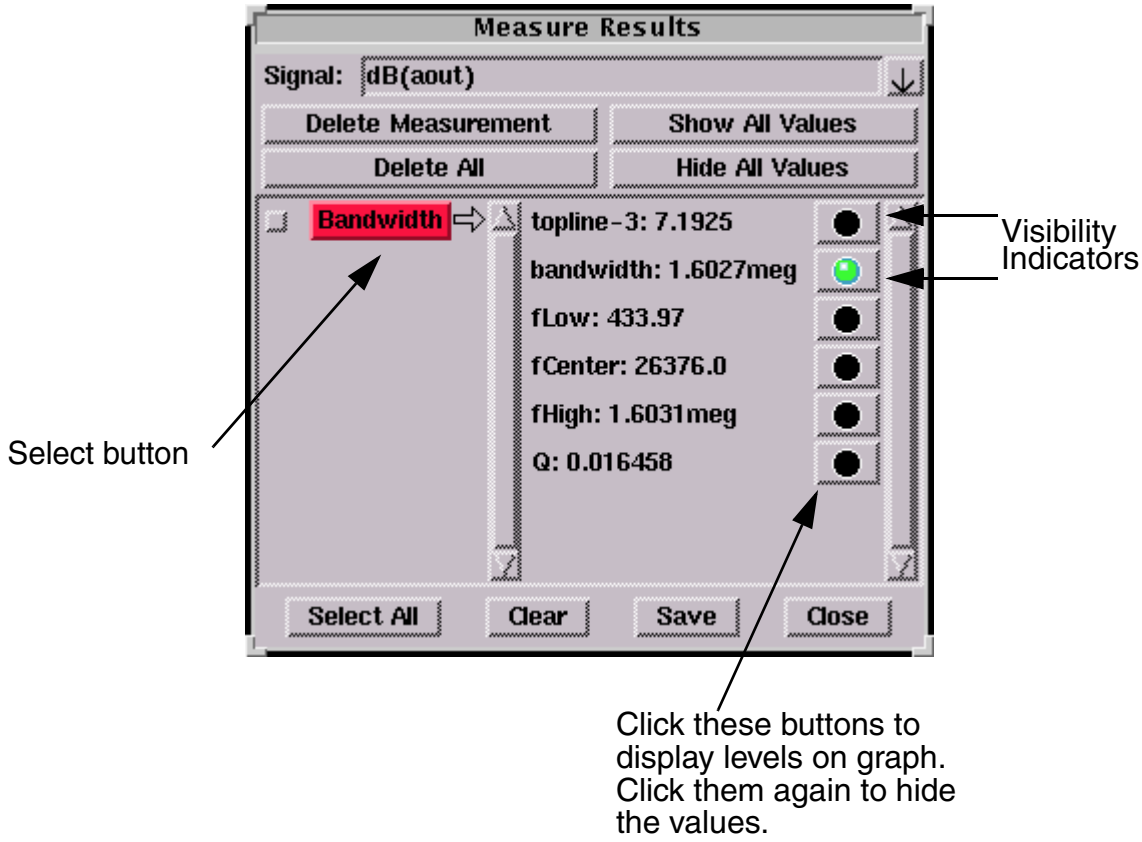
   c. You can select the measurement labels and move them if the graph becomes too cluttered. Position the cursor over the text. Then left-click and hold while moving the cursor to a new location.

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 **Signal > Measure Results...**. The 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, choose **File > Exit**.

Viewing HSPICE Results

This tutorial explains how to use CosmosScope to view the analysis results from the simulation of a single-stage amplifier design.

This section covers the following topics:

- Setting up the Design Data
- Viewing HSPICE Transient Analysis Waveforms
- Viewing AC Analysis Waveforms
- Performing Measurements on an 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/CScope/hspice_amp` directory to the synopsys_tutorial directory:

   UNIX:

   ```
   cp -r install_home/examples/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\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 the v(aout) signal 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 right-click the mouse button and choose **Signal > Delete Signal** from the pop-up menu that appears. Do this for each signal you want to delete.
8. Look at the vdb(aout) (dBV):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

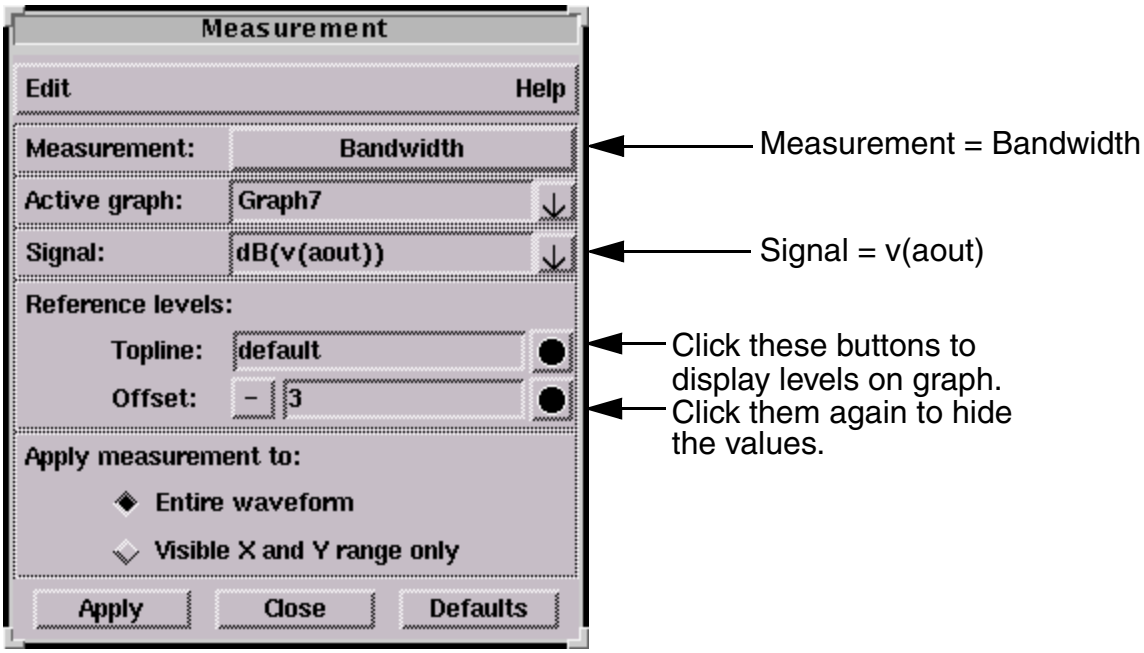
The Measurement Tool within CosmosScope provides a method of performing various measurements on a waveform. You check the bandwidth and gain of the single-stage amplifier output signal v(aout) as follows:

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.

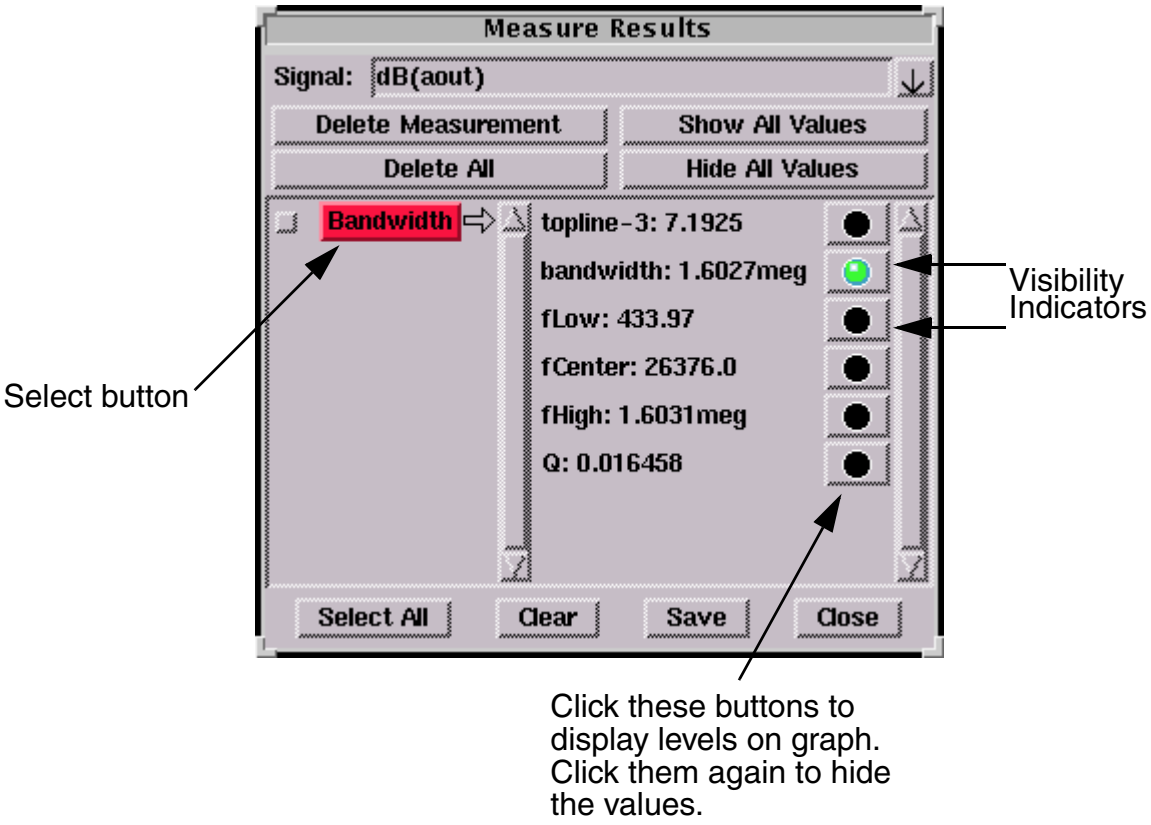
   c. You can select the measurement labels and move them if the graph becomes too cluttered. Position the cursor over the text. Then left-click and hold while moving the cursor to a new location.

5. You can get more information about each of the measures 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 cursor to the v(aout) signal name.

b. Use the popup menu and choose the **Signal Menu > Measure Results...** item. The 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 right column, the different values 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.

3

Signals

Explains how to convert, pan, and change signals and their associated properties. Also explains about how to convert and combine analog and digital signals.

This section covers the following topics:

- Converting Analog Signals to Digital
- Converting Digital Signals to Analog
- Converting Frequency Domain Signals to the Steady-State Time Domain
- Changing Signal Appearance
- Changing Signal Zoom Level
- Panning Signals
- Changing Signal Display Colors
- Defining Custom Signal Display Colors
- Moving Signals to Graph Regions
- Specifying Analog Signal Fill Patterns
- Combining Digital Signals into a Bus
- Displaying the Digital Signal Grid
- Changing Symbol Styles
- Changing Signal Colors

Converting Analog Signals to Digital

To convert analog signals to digital:

1. Select one or more analog signals that are currently graphed. You can click either the name of the signal or the signal attributes line just below the signal name to select a signal.
2. Choose **Signal > Convert to Digital (A2D)...** from the main menu.
3. Select a single- or multiple-bit conversion, and enter the necessary threshold values. You can also enter a value for sample period size.
4. Click **Ok** to convert the signal(s).

Converting Digital Signals to Analog

This section covers the following topics:

- Converting Single-Bit Digital Signals to Analog
- Converting Digital Bus Signals to Analog

Converting Single-Bit Digital Signals to Analog

To convert single-bit digital signals to analog:

1. Select one or more single-bit digital signals that are currently graphed. You can click either the name of the signal or the signal attributes line just below the signal name to select a signal.
2. Choose **Signal > Convert to Analog (D2A)...** from the main menu.
3. Enter values for the digital 0 and 1 states (VLow and VHigh, respectively).
4. Enter values for the rising and falling time (TRise and TFall), which specifies the rising and falling time when any value changes in the input digital signal.
5. If needed, select **Ignore glitches shorter than** or **Convert to current, Resistance**, and enter the necessary values.
6. Select an option to convert uknown values. If you select **Last known** and the first state of the digital signal is unknown, **(VLow+VHigh)/2** is used.
7. Click **Ok** to convert the signal(s).

Converting Digital Bus Signals to Analog

To convert digital bus signals to analog:

1. Select one or more digital bus signals that are currently graphed. You can click either the name of the signal or the signal attributes line just below the signal name to select a signal.
2. Choose **Signal > Convert to Analog (D2A)...** from the main menu.
3. Enter the minimum and maximum values for the converted signals (Lower bound and Upper bound).
4. Enter values for the rising and falling time (TRise and TFall), which specifies the rising and falling time when any value changes in the input digital signal.
5. If needed, select **Ignore glitches shorter than** or **Convert to current, Resistance**, and enter the necessary values.
6. Click **Ok** to convert the signal(s).

Converting Frequency Domain Signals to the Steady-State Time Domain

Select one or more frequency domain signals that are currently graphed, and choose **Signal > To Time Domain** from the main menu.

Changing Signal Appearance

Double-click the signal name, which opens the Signal Attributes dialog box.

Several of the attributes in the Signal Attributes dialog box are also available from the **Signal** menu.

Changing Signal Zoom Level

Select a signal, and press the **z** key to zoom in or **Shift+z** to zoom out. Press the **f** key to zoom in to fit the size of the current window. All signals on a graph change at the same zoom level.

You can also change zoom levels using any of the following other methods:

- Click one of the zoom icon button at the top of the main window
- Choose **Graph > Zoom** from the main CosmosScope menu.
- Click and hold the left mouse button, then drag your mouse to create a frame around the area you want 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 on the selected axis.
- Click and hold the right mouse button over one of the axes or over an axis name in the legend, and choose **Zoom > <zoom level>** from the pop-up menu that appears.

Panning Signals

To pan a signal, right-click one of the axes and choose **Pan > Right/Left/Up/Down** from the pop-up menu that appears.

You can also pan signals using one of the following methods:

- Click one of the axes, and choose **Graph > Pan > Right/Left/Up/Down** from the main CosmosScope menu.
- 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.
- Right-click the x-axis, and choose **Attributes...** from the pop-up menu that appears. Move the Slider bar in the Range field to pan the view left and right.

Changing Signal Display Colors

Select one or more signals, choose **Signal > Color** from the main menu, and select one of the available colors.

You can also define custom colors from the Signal tab, which is located in the CosmosScope preferences (**Edit > Preferences...**). See Defining Custom Signal Display Colors for more information.

Defining Custom Signal Display Colors

This section covers the following topics:

- Defining Analog Signal Colors
- Defining Digital Signal Colors

Defining Analog Signal Colors

Note: If your screen colors are mapped to Mono, signals are displayed as a variety of dashed lines. These dashed lines cannot be customized.

To define custom signal display colors for analog signals:

1. Choose **Edit > Preferences...**, and click the Signal tab.
2. Click the **Add** button. A new column appears at the end of Map1, Map2, and Mono rows.
3. Click the new Map1 color button. The Color Editor appears.
4. Click the **Define Custom Colors** button.
5. Move the cursor in the colored box to select a color.
6. Move the vertical slide bar to adjust the brightness of the selected color.
7. Click the **Add to Custom Colors** button to add your defined color to the Custom Colors pallette.

8. Once you are finished defining custom colors, click **Ok** to close the Color Editor.
9. Click the **Save** button at the bottom of the Signal tab to save your changes.

Defining Digital Signal Colors

Note: If your screen colors are mapped to Mono, signals are displayed as a variety of dashed lines. These dashed lines cannot be customized.

To define custom digital signal display colors:

1. Choose **Edit > Preferences...**, and click the Signal tab.

   For digital signals, CosmosScope displays different colors and line styles for different logical states. Currently, CosmosScope supports logic_4, std_logic, and nanosim_logic type digital signals.
2. Choose a logic type for which you want to define custom colors.
3. Click a button in the Colors row that you want to change. The Color Editor appears.
4. Click the **Define Custom Colors** button.
5. Move the cursor in the colored box to select a color.
6. Move the vertical slide bar to adjust the brightness of the selected color.
7. Click the **Add to Custom Colors** button to add your defined color to the Custom Colors pallette.
8. Once you are finished defining custom colors, click **Ok** to close the Color Editor.
9. Click the **Save** button at the bottom of the Signal tab to save your changes.

Moving Signals to Graph Regions

Select one or more signals, and choose one of the options available from **Signal > Move to Stack Region**:

- **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.

Specifying Analog Signal Fill Patterns

Select one or more analog signals on a graph, and choose one of the available fill patterns from **Signal > Bar**. The area under the signal curve fills in with the pattern you select.

Combining Digital Signals into a Bus

To combine digital signals from the trace graph region into a single digital bus:

1. Select two or more digital signals from the trace graph region.

   The first signal you select is considered the least significant bus (LSB), and each additional signal you select is considered more significant than the previous signal. The last signal you select is assumed to be the most significant bus (MSB).

2. Choose **Signal > Create Bus** from the main menu. The signal is plotted in CosmosScope and given the designation Groupnumber.

Displaying the Digital Signal Grid

Select a signal name in a graph window for which you want to display the grid, and choose **Signal > Display Digital Signal Grid**.

Changing Symbol Styles

Select one or more signal names on a graph, and choose **Signal > Symbol**.

Changing Signal Colors

Select one or more signal names on a graph, and choose **Signal > Color**.

4

Graphs

Explains how to display, save, and use graphs in CosmosScope. Also explains how to annotate and export the graphs.

This section covers the following topics:

- Displaying Graphs
- Saving Graphs
- Opening Outlines
- Saving Outlines
- Placing Trace Markers
- Changing Trace Height Display
- Annotating Graphs
- Exporting Images
- Changing Graph Font Face and Style
- Changing Graph Zoom Level
- Displaying the Slider Bar

Displaying Graphs

To graph waveforms on the work surface, use the Signal Manager. See the *CosmosScope Tools Reference* for more information on the Signal Manager.

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

To update the waveform dynamically, choose **Edit > Preferences...** . In the Scope Preferences form, select the Display tab. Select the Dynamic Waveform Display button and enter the interval, in seconds, desired for dynamically updating the displayed waveform.

Saving Graphs

To save a graph:

1. Choose **File > Save Graph...** from the CosmosScope main menu.
2. Specify the name and location for the graph to be saved, and click **Save**. The Save Graph dialog box opens.

   Select one of the following options:

   - Save copy of waveforms with graph

     This option saves copies all of the related plotfiles in the specified directory. These plotfiles are separate from the originals and are not overwritten by subsequent analyses.

   - Reference the current plot files

     This option maintains the connection to the original plotfiles. The graph can be automatically updated when reopened if you specify any Replace or Append plot actions for an analysis. Make sure you specify either absolute or relative paths to the graph plotfiles.

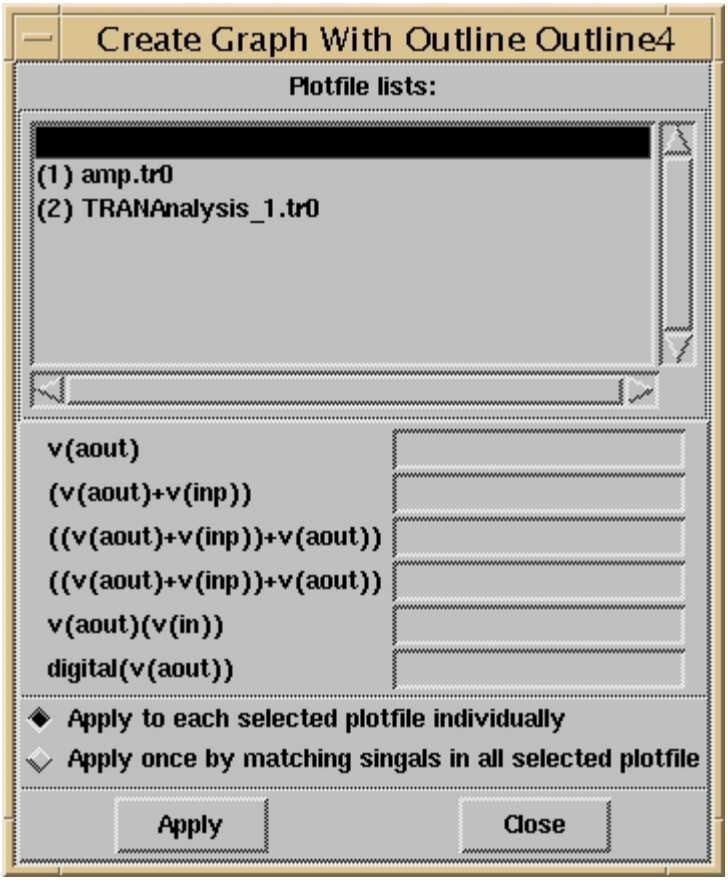
3. Click the **Save** button.

Opening Outlines

To open a saved outline:

1. Choose **Open > Outlines...** from the CosmosScope main menu.

2. Select an outline file, and click the **Open** button. The Create Graph With Outline dialog box appears.



3. Specify any desired waveforms to open in the middle section of the Create Graph With Outline dialog box.

   You can fill in the waveform entries using one of the following methods:

   - Paste waveforms into the list by selecting them in a graph or plotfile window, placing the cursor in the text box next to the placeholder, and then clicking the middle mouse button. You cannot type in the entry boxes.
   - Select one or more plotfiles in the Plotfile lists box, which lists all the opened plotfiles. CosmosScope searches the selected plotfiles for waveform names and fills the entries with corresponding waveforms. Selecting the empty line in the Plotfile lists box clears all the entries.

When multiple plotfiles are selected, clicking the **Apply to each selected plotfile individually** radio button creates an outline for each selected plotfile; clicking the **Apply once by matching signals in all selected plotfile** radio button creates an outline only for waveforms that are located in multiple plotfiles.

4. Click the **Apply** button to open a new graph window and plot the waveforms.

Saving Outlines

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:

1. Choose **File > Save Outline...** from the main CosmosScope menu.
2. Specify the name and location for the graph outline to be saved, and click **Save**. The Graph Outline dialog box opens.
3. Select all desired attributes that you want to include in the graph outline.

   Clicking the **Dependencies** check box maintains the connection to the plotfile. The outline can be automatically updated when reopened if you specify any Replace or Append plot actions for an analysis.
4. Click the **OK** button to save the graph outline.

Placing Trace Markers

To place a Trace marker:

1. Click and hold the New Marker Symbol, then drag the marker to the desired location.
2. Release the mouse button to place the marker. The logic state of the location is displayed in the Marker field.

A Trace graph uses the same pop-up menus available in the Analog graph region with the addition of the Trace popup menu. The Trace pop-up menu operates only in the Trace graph region.

Changing Trace Height Display

To change the trace height display:

1. Open the CosmosScope preferences (**Edit > Preferences...**).
2. Click the XY tab.
3. In the "Digital region options" section located at the top of the XY tab, change the analog or digital trace height values as necessary.

Annotating Graphs

Choose **Graph > Annotate Info...**, and click the box next to the text variable(s) you want to add. The information that you add appears on the graph in the active graph window, which you can then move to any position on the graph.

Exporting Images

To export an image of the active graph window:

1. Choose **File > Export Image...**.
2. Choose an output image file type.
3. Save the image.

Changing Graph Font Face and Style

Choose **Graph > Font...** from the main menu, and select the desired font, font style, or size.

Changing Graph Zoom Level

See Changing Signal Zoom Level for information on changing zoom levels.

Displaying the Slider Bar

To display the slider bar on the x-axis, right-click the x-axis, and choose **X Axis Slider** from the pop-up menu that appears.

You can also display the slider bar using one of the following methods:

- Right-click the x-axis, and choose Attributes... from the pop-up menu that appears. Then, click the **X axis slider** radio button.
- Click the x-axis, and choose **Axis > Attributes...** from the CosmosScope main menu. Then, click the **X axis slider** radio button.

5

Waveforms

Explains how to compare waveforms and create multi-member parameter files and multi-member waveforms.

This section covers the following topics:

- Comparing Waveforms
- Creating Multi-Member Parameter Files
- Merging All or Selected Plotfile Waveforms into Multi-Member Waveforms
- Creating Multi-Member Waveforms

Comparing Waveforms

To compare waveforms:

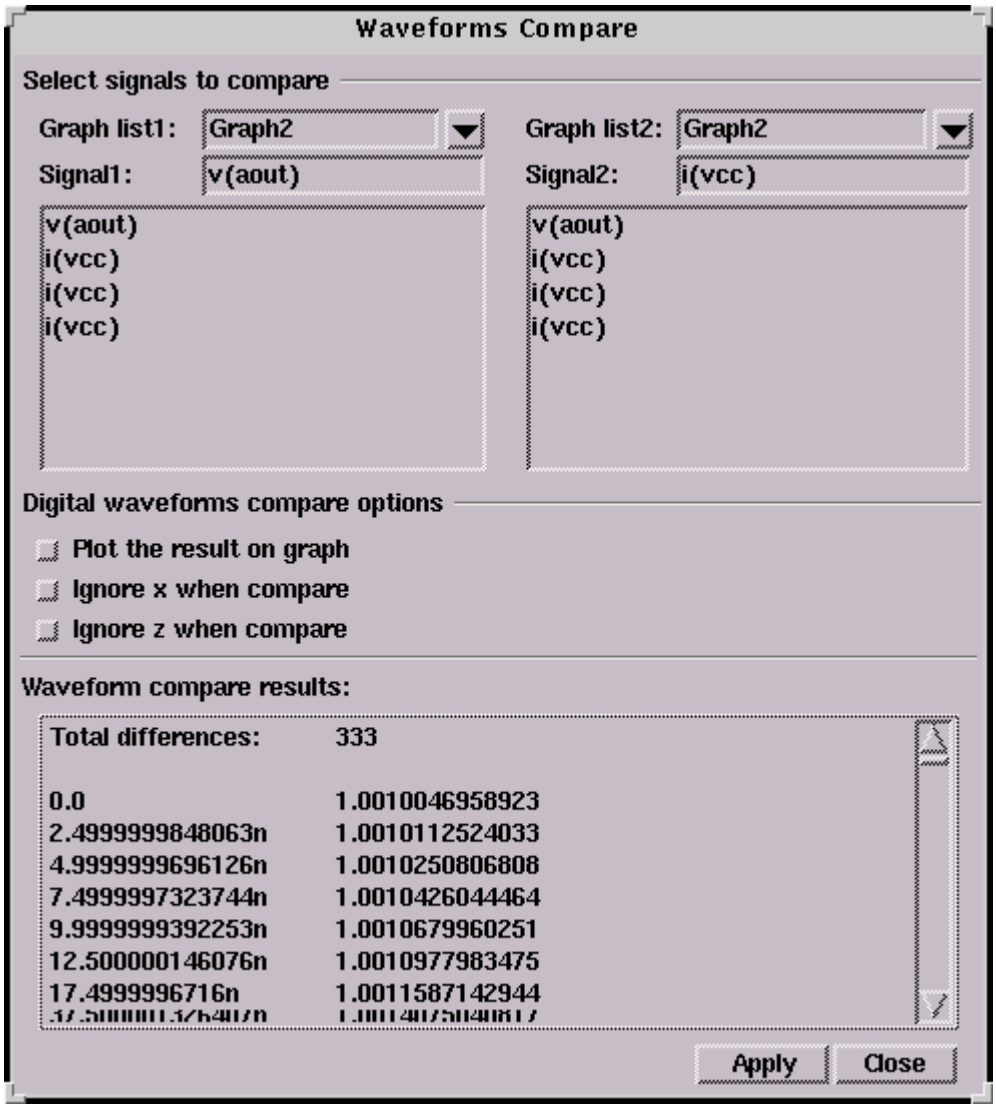
1. Choose **Graph > Waveform Compare...** from the main menu.
2. Select the graph that contains the first signal you want to compare from the Graph list1 list. The signals that appear on the graph are listed in the field just below the Signal1 field.
3. Select the signal name that you want to compare. The signal name appears in the Signal1 field.

   If you want to see more information about the signal, move your mouse over the signal name. The following information appears in the status bar at the bottom of the CosmosScope main window:

   - full name of the signal
   - graph on which the signal is located
   - signal handle

- waveform handle
- plotfile name

4. Repeat steps 2 and 3 to select a signal for Signal2, which might be from a different Graph.
5. If the signals are digital, specify any desired compare options.



6. Click **Apply** to generate the result.

Creating Multi-Member Parameter Files

Before merging any waveforms into multi-member waveforms, you might consider defining the list of parameters to use when creating the multi-member waveform plotfile instead of using the defaut "run" parameters. See Creating Multi-Member Parameter Files for more information.

To create a *.par file:

1. Open a new text file, and enter the desired parameters to use when the multi-member waveform is created. Each parameter is entered on its own line. For example:

   ```
   vdd = 1.2
   temp = -27
   mos = 'ff'
   ```

2. Save the text file, and name it using the <plot_filename>.par syntax. The <plot_filename> is the same as the multi-member file name you are going to create. Save the *.par file in the same directory as the plot files you are using to create the multi-member waveform.
3. Merge the plotfiles into multi-member waveforms, and ensure the default "run" parameter is used. The plotfiles are merged using the parameter file you created instead of using the "run" parameter.

   See Merging All or Selected Plotfile Waveforms into Multi-Member Waveforms to continue.

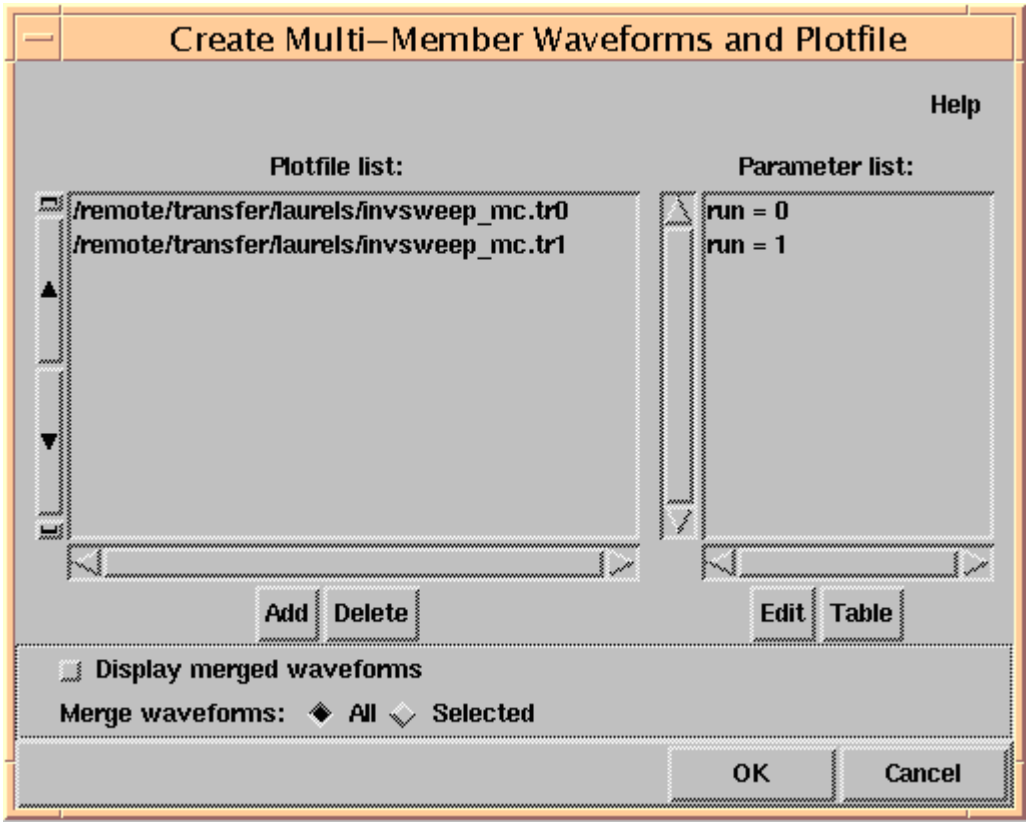
Merging All or Selected Plotfile Waveforms into Multi-Member Waveforms

Before merging any waveforms into multi-member waveforms, you might consider defining the list of parameters to use when creating the multi-member waveform plotfile instead of using the defaut "run" parameters. See Creating Multi-Member Parameter Files for more information.

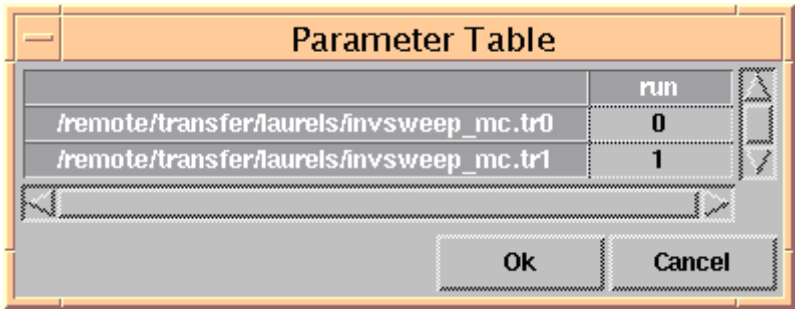
To merge all or selected plotfile waveforms into multi-member waveforms:

1. Choose **File > Merge Plotfiles...** . The Select Plotfiles to Merge dialog box appears. If you previously selected a plot file to merge, the last plot file selection is automatically selected in the File names: text field.

2. Select the plotfiles you want to include in the multi-member waveform and click the **Open** button. The Create Multi-Member Waveforms and Plotfile dialog box appears with the selected files in the Plotfile List field.



If you want to see a table containing a list of plotfiles vs. the parameters, click the **Table** button. The following Parameter Table dialog box opens:

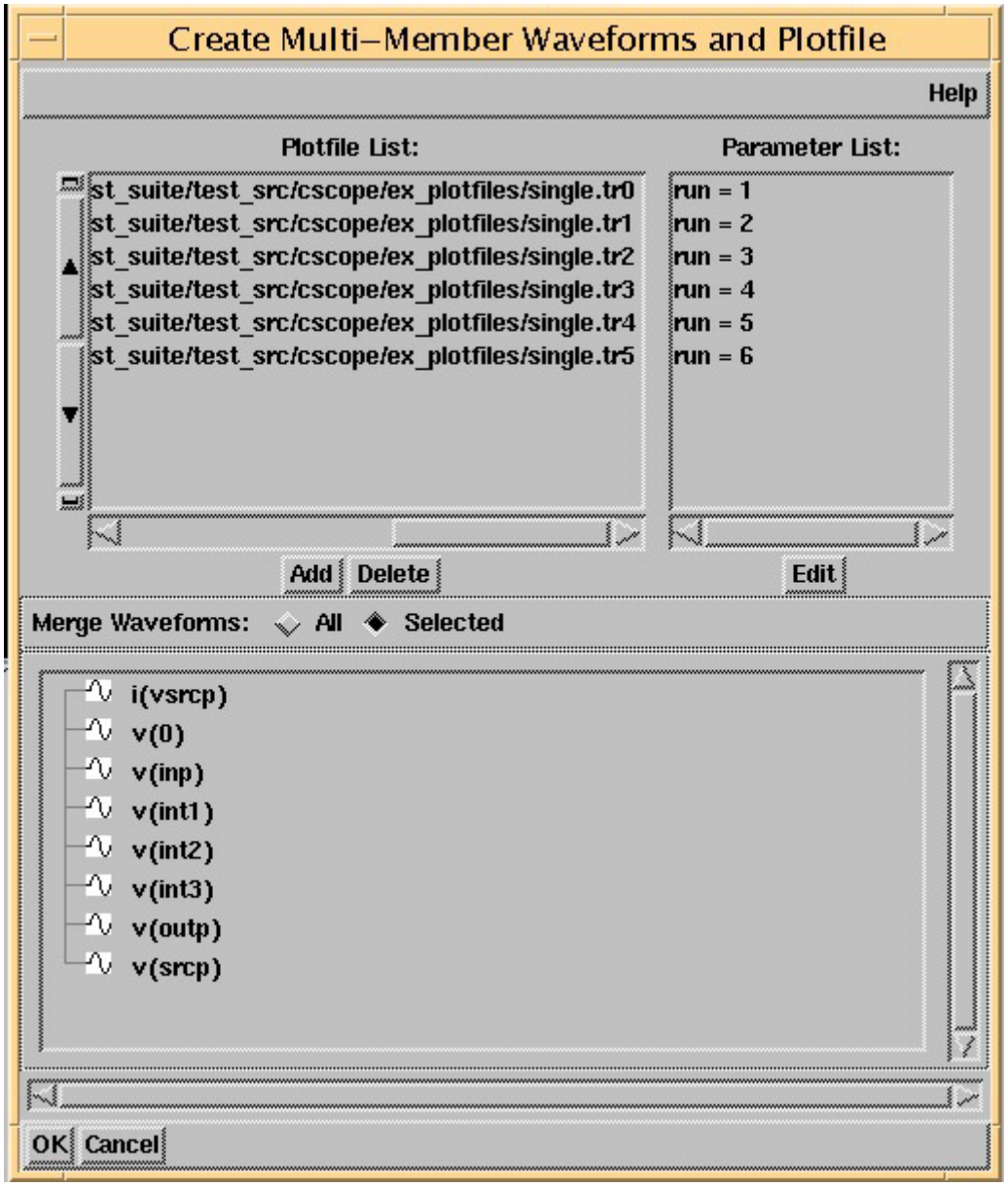


You can use the Parameter Table to swap values between plotfiles or assign a particular set of parameter values to a specific plotfile. Clicking a value in the run column opens a pop-up menu that allows you to choose from available parameter values for each plotfile. Clicking **OK** checks that the

parameter sets are unique and redisplays the new parameter list in the Create Multi-Member Waveform dialog box. Only the parameter sets that correspond to a plotfile or waveform are displayed. All other parameter sets are removed from the Parameter List.

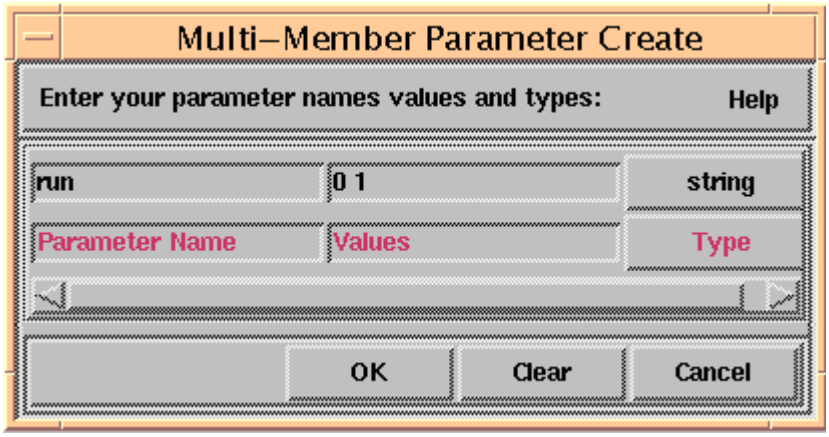
3. If you want to merge all waveforms, ensure the **All** radio button is selected next to the Merge Waveforms field, and skip to step Step 6. If you want to merge only selected waveforms, continue to the next step.

4. Click the **Selected** radio button next to the Merge Waveforms text.



5. Select the desired waveforms to merge.

6. Click the **Edit** button under the Parameter List window. The Multi-Member Parameter Create dialog box appears.



7. Enter the desired values for the following fields:
   - Parameter Name -- The name of the parameter. The default is “run”.
   - Values -- A list of values the parameters can have.
   - Type -- The type of parameter. You can select string, number, or set. The default is “string”.

   For example:

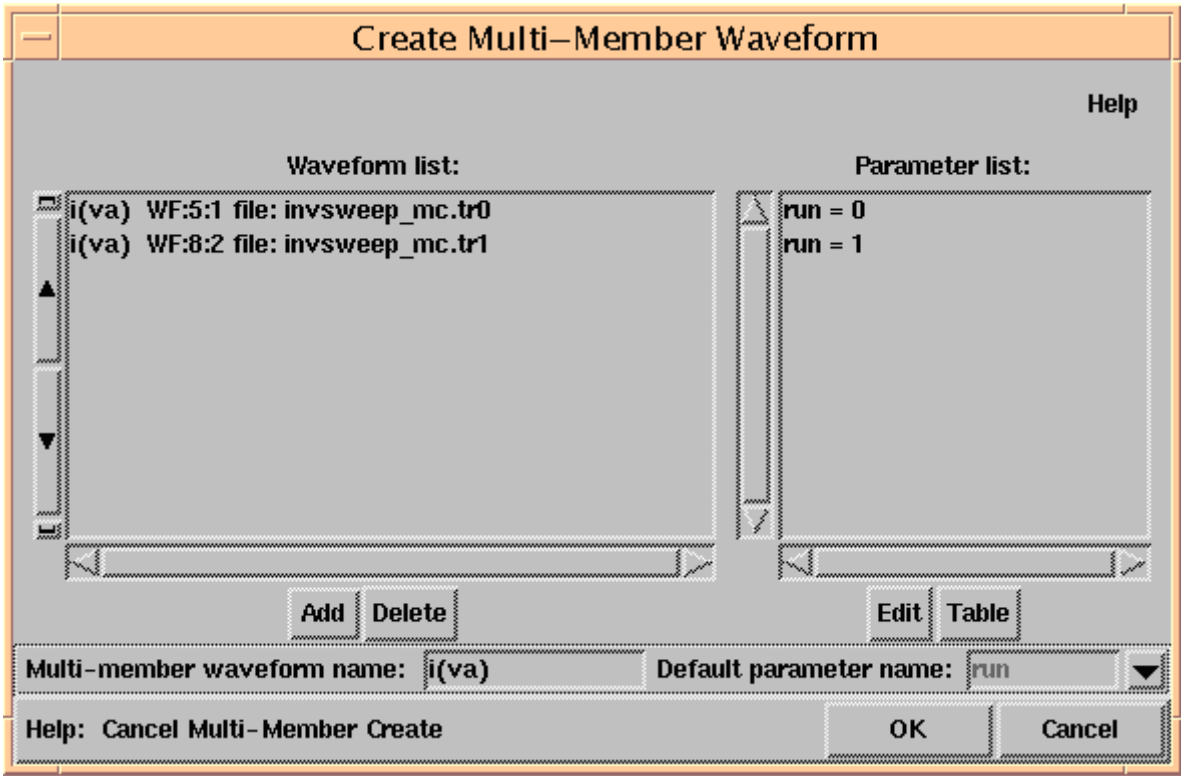
   Parameter Name: run Values: 1 2 3 4 Type: number

8. Click the **OK** button to save your settings. The Multi-Member Parameter Create dialog box closes.
9. Select the **Display Merged Waveforms** check box to plot all merged plotfiles to the graph window. By default, this check box is unchecked, and the merged plotfiles are not plotted to the graph window.
10. Click the **OK** button in the Create Multi-Member Waveforms and Plotfile dialog box to merge the waveforms. The multi-member waveform appears and you are prompted to save the file.

Creating Multi-Member Waveforms

To merge waveforms into a single multi-member waveform:

1. Choose **Signal > Create Multi-Member...** . The Create Multi-Member Waveform dialog box appears with the selected waveform(s) in the Waveform List field.
2. Click the **Add** button to add waveforms to the Waveform List, or click the **Delete** button to delete waveforms from the Waveform List.

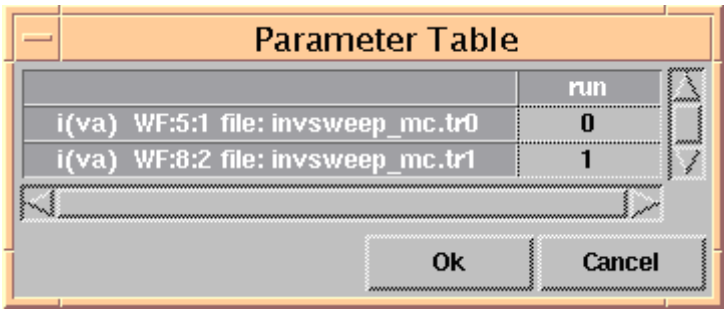


The Default parameter name drop-down menu is set to run (by default). You can choose one of the following default parameter names:

run *number*	Selected by default if the merged signals merged have the same name and come from different plotfiles. The *number* is from the extension of the file (for example, the *.tr0 and *.tr1 file extensions result in run numbers 0 1). If the file extension does not contain a number or a duplicate number exists, the run number is assigned based on the total number of merged signals.

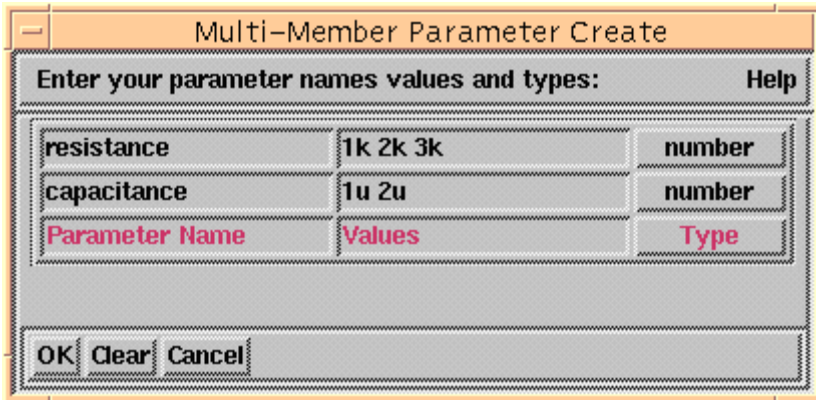
signal *name*	Selected by default if different signal names are merged from a single plotfile. The *name* is the name of the signal.
run_signal *number name*	Selected by default if different signal names are merged from different plot files. The *name* is the signal name, and the number is either the number stripped from the file extension or a count of the total number of merged signals.
custom *name*	Displays the parameter name and values defined using the Parameter Create dialog box. If you do not change the default parameter, custom is assigned to the default name.

If you want to see a table containing a list of waveforms vs. the parameters, click the **Table** button. The following Parameter Table dialog box opens:



You can use the Parameter Table to swap values between waveforms or assign a particular set of parameter values to a specific waveform. Clicking a value in the run column opens a pop-up menu that allows you to choose from available parameter values for each waveform. Clicking **OK** checks that the parameter sets are unique and redisplays the new parameter list in the Create Multi-Member Waveform dialog box. Only the parameter sets that correspond to a plotfile or waveform are displayed. All other parameter sets are removed from the Parameter List.

3. To modify parameters in the Parameter List window, click the **Edit** button under the Parameter List window. The Multi-Member Parameter Create dialog box appears.



4. Enter the desired values for the following fields:
   - Parameter Name -- The name of the parameter. The default is “run”.
   - Values -- A list of values the parameters can have.
   - Type -- The type of parameter. You can select string, number, or set. The default is “string”.

   For example:

   Parameter Name: run Values: 1 2 3 4 Type: number
5. Click the **OK** button to save your settings. The Multi-Member Parameter Create dialog box closes.
6. Click the **OK** button in the Create Multi-Member Waveform dialog box to merge all the selected waveforms from the Waveform List into a single multi-member waveform. CosmosScope displays the final merged waveform and does not prompt you to save the merged waveform.

Index

A

AimAppCom package, using to communicate with CosmosScope 3
analog signals
  converting to digital 24
  specifying fill patterns 29

B

buses
  combining digital signals into 29
  converting to analog 25

C

colors, changing for signal display 27

D

digital signal grid, displaying 29
digital signals
  buses
    combining into 29
    converting to analog 25
  converting to analog 24
  grid, displaying 29
  single-bit, converting to analog 24
  trace height display, changing 35

F

fill patterns, specifying for analog signals 29
fonts, changing in graphs 35
frequency domain, converting to steady-state domain 25

G

graphs
  annotating 35
  changing font style 35
  displaying 31
  exporting images from 35
  saving 32
  saving outlines from 34

I

images, exporting 35
invoking CosmosScope
  on UNIX platforms 2
  on Windows platforms 1

M

multi-member waveforms
  creating 44
  creating parameter files for 39
  merging from plotfiles 39

O

outlines
  opening 32
  saving 34

P

panning signals 26
pareto analysis, performing 8
Performing Measurements on a Waveform 14
Performing Measurements on an HSPICE Waveform 20
plotfiles
  opening 5

S

Setting up the Design Data 17
Setting up the Saber Simulation Data 11
signals
  analog
    converting to digital 24
    fill patterns, specifying 29
  appearance, changing 25
  changing colors 27
  digital
    buses, combining into 29

buses, converting to analog 25
converting to analog 24
signal grid, displaying 29
single-bit, converting to analog 24
trace height display, changing 35
graph regions, moving to 28
panning 26
steady-state domain, converting to 25
symbol styles, changing 30
zoom level, changing 26
steady-state domain, converting to frequency domain 25
symbol styles, changing 30

T

third-party tools
connecting to CosmosScope 3
testing connection to CosmosScope 4
trace height display, changing 35
trace markers, placing 34

V

Viewing AC Analysis Waveforms 18
Viewing HSPICE Transient Analysis Waveforms 17
Viewing Saber AC Analysis Waveforms 13
Viewing Saber Transient Analysis Waveforms 12

W

waveforms
comparing 37
multi-member
creating 44
creating parameter files for 39
merging from plotfiles 39