

# **Waveform Calculator User Guide**

**Product Version 5.0**  
**October 2003**

© 1999-2003 Cadence Design Systems, Inc.  
Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 1-800-862-4522.

All other trademarks are the property of their respective holders.

**Restricted Print Permission:** This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

**Restricted Rights:** Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

# Contents

## 1

<u>Overview</u>	9
<u>About the Calculator</u>	10
<u>About the Algebraic and RPN Modes</u>	10
<u>Using Keys</u>	11
<u>Entering Constants</u>	11
<u>About Standard and RF Modes</u>	12
<u>Window Size and Location</u>	12
<u>Calculator Bindkey</u>	12
<u>Starting the Calculator</u>	12
<u>Closing and Quitting the Calculator</u>	13

## 2

<u>Using the Calculator</u>	15
<u>Selecting Data</u>	15
<u>Selecting Data in a Schematic Window</u>	16
<u>Selecting Curves in the Waveform Window</u>	18
<u>Selecting Simulation Results with the Results Browser</u>	19
<u>Using the Results Browser</u>	20
<u>Starting the Browser</u>	22
<u>Managing the Display</u>	24
<u>Expanding Nodes</u>	26
<u>Plotting Waveforms</u>	27
<u>Copying Expressions into the Calculator</u>	27
<u>Deleting UNIX Files</u>	28
<u>Renaming UNIX Files</u>	28
<u>Displaying Properties</u>	29
<u>Using External Data</u>	30
<u>Selecting Results</u>	30
<u>Plotting or Printing Results</u>	30
<u>Plotting Expressions</u>	31

## Waveform Calculator User Guide

---

<u>Printing One Expression Value</u>	31
<u>Printing a Range of Expression Values</u>	32
<u>Setting the Range Calculation Increment</u>	33
<u>Applying Waveform Transformations</u>	34
<u>Shifting the X Axis (L shift)</u>	34
<u>Y Axis Flip</u>	35
<u>Using Memories</u>	36
<u>Storing Memories</u>	36
<u>Recalling Memories</u>	37
<u>Deleting Memories</u>	38
<u>Saving and Loading Memories</u>	38
<u>Defining Functions and Function Keys</u>	39
<u>Defining New Functions</u>	39
<u>Assigning Function Keys</u>	41

### 3

<u>RPN Mode</u>	43
<u>About the Buffer</u>	43
<u>Using the Keypad, Functions, and Memories</u>	43
<u>Entering Variables</u>	44
<u>Entering Constants</u>	44
<u>Entering Multiple Expressions</u>	44
<u>Evaluating the Buffer</u>	45
<u>About the Stack</u>	45
<u>Displaying the Stack</u>	45
<u>Moving the Buffer and Stack Register</u>	46
<u>Pushing Expressions on the Stack</u>	46
<u>Exchanging the Buffer and Stack</u>	47
<u>Recalling the Last Expression</u>	47
<u>Clearing the Buffer and Stack</u>	48
<u>Operators and Functions</u>	48
<u>Single-Expression Functions</u>	49
<u>Two-Expression Functions and Operators</u>	50
<u>Trigonometric Functions</u>	51
<u>Special Functions</u>	51

## Waveform Calculator User Guide

---

<u>Example: Average Value of a Current</u>	52
<u>RPN Mode Special Functions</u>	52
<u>Average Function</u>	52
<u>Bandwidth Function</u>	53
<u>Clip Function</u>	56
<u>Compression Function</u>	56
<u>CompressionVRI Function</u>	58
<u>Convolution (convolve) Function</u>	59
<u>Threshold Crossing (cross) Function</u>	60
<u>dBm Function</u>	61
<u>Delay Function</u>	61
<u>Derivative (deriv) Function</u>	62
<u>Discrete Fourier Transform (dft) Function</u>	62
<u>eyeDiagram Function</u>	66
<u>Flip Function</u>	67
<u>Fourier Evaluation (fourEval) Function</u>	67
<u>Frequency Function</u>	68
<u>Gain (gainBwProd/gainMargin) Functions</u>	68
<u>Group Delay Function</u>	68
<u>Harmonic Function</u>	68
<u>Harmonic Frequency Function</u>	69
<u>iinteg Function</u>	70
<u>ipn Function</u>	70
<u>ipnVRI Function</u>	71
<u>Lshift Function</u>	73
<u>Minimum and Maximum Functions</u>	74
<u>Overshoot Function</u>	76
<u>Phase Margin Function</u>	77
<u>Phase Noise Function</u>	77
<u>Power Spectral Density (psd) Function</u>	79
<u>Power Spectral Density Baseband (psdbb) Function</u>	80
<u>Rise Time Function</u>	82
<u>Root-Mean-Square (rms) Function</u>	83
<u>Root-Mean-Square (rms) Noise Function</u>	83
<u>Root Function</u>	83
<u>Sample Function</u>	84

## Waveform Calculator User Guide

---

<u>Settling Time Function</u>	85
<u>Slew Rate Function</u>	86
<u>Spectral Power Function</u>	86
<u>Standard Deviation (stddev) Function</u>	87
<u>Table Function</u>	87
<u>Tangent Function</u>	88
<u>Total Harmonic Distortion (thd) Function</u>	89
<u>Value Function</u>	90
<u>X Value (xval) Function</u>	90

## 4

<u>Algebraic Mode</u>	93
<u>Operators and Functions</u>	93
<u>Single-Expression Functions</u>	93
<u>Two-Expression Functions and Operators</u>	94
<u>Trigonometric Functions</u>	95
<u>Special Functions</u>	96
<u>Example: Average Value of a Current</u>	97
<u>Example: Delay Function</u>	97
<u>Algebraic Mode Special Functions</u>	99
<u>Average Function</u>	99
<u>Bandwidth Function</u>	99
<u>Clip Function</u>	102
<u>Compression Function</u>	103
<u>CompressionVRI Function</u>	104
<u>Convolution (Convolve) Function</u>	105
<u>Threshold Crossing (cross) Function</u>	106
<u>dBm Function</u>	108
<u>Delay Function</u>	108
<u>Derivative (deriv) Function</u>	109
<u>Discrete Fourier Transform (dft) Function</u>	109
<u>eyeDiagram Function</u>	114
<u>Flip Function</u>	114
<u>Fourier Evaluation (fourEval) Function</u>	114
<u>Frequency Function</u>	115

## Waveform Calculator User Guide

---

<u>Gain (gainBwProd/gainMargin) Functions</u>	115
<u>Group Delay Function</u>	116
<u>Harmonic Function</u>	116
<u>Harmonic Frequency Function</u>	117
<u>iinteg Function</u>	117
<u>ipn Function</u>	118
<u>ipnVRI Function</u>	119
<u>Lshift Function</u>	122
<u>Minimum and Maximum Functions</u>	122
<u>Overshoot Function</u>	125
<u>Phase Margin Function</u>	126
<u>Phase Noise Function</u>	126
<u>Power Spectral Density (psd) Function</u>	128
<u>Power Spectral Density Baseband (psdbb) Function</u>	129
<u>Rise Time Function</u>	130
<u>Root-Mean-Square (rms) Function</u>	132
<u>Root-Mean-Square (rms) Noise Function</u>	132
<u>Root Function</u>	133
<u>Sample Function</u>	133
<u>Settling Time Function</u>	134
<u>Slew Rate Function</u>	135
<u>Spectral Power Function</u>	136
<u>Standard Deviation (stddev) Function</u>	137
<u>Table Function</u>	137
<u>Tangent Function</u>	138
<u>Total Harmonic Distortion (thd) Function</u>	139
<u>Value Function</u>	141
<u>X Value (xval) Function</u>	141

## 5

<u>RF Mode</u>	143
<u>Calculator Buttons for RF</u>	147
<u>Special Function Buttons for RF</u>	149

## A

### Using the Calculator Special Functions with SpectreRF

<u>Simulation Results</u> .....	153
<u>Using the ipn and ipnVRI Calculator Special Functions</u> .....	153
<u>Setting Up the Simulation Environment</u> .....	154
<u>Editing the Schematic</u> .....	158
<u>Setting Up the Analyses</u> .....	161
<u>Selecting Outputs to Save</u> .....	167
<u>Running the Simulation</u> .....	167
<u>Plotting the IP3 Curves Using the Direct Plot Form</u> .....	167
<u>Plotting IP3 Using the Calculator ipn and ipnVRI Special Functions in the RPN Mode</u> .	171
<u>Plotting IP3 Using the Calculator ipn and ipnVRI Special Functions in the Algebraic Mode</u>	177
<u>Using the IPN Special Function with Scalar Inputs</u> .....	182
<u>Setting Up the Simulation Environment</u> .....	182
<u>Setting Up the Analyses</u> .....	182
<u>Edit the prf Design Variable</u> .....	186
<u>Running the Simulation</u> .....	187
<u>Plotting the Single Point IP3 Using the Direct Plot Form</u> .....	187
<u>Using the spectralPower Special Function</u> .....	191
<u>Setting Up the Simulation Environment</u> .....	191
<u>Setting Up the PSS Analysis</u> .....	191
<u>Selecting Outputs to Save</u> .....	195
<u>Running the Simulation</u> .....	196
<u>Plotting the Power Spectrum Using the Direct Plot Form</u> .....	196
<u>Plotting the Power Spectrum Using the Calculator spectralPower Special Function</u>	199

---

## Overview

The Waveform Calculator is a scientific calculator with both algebraic and Reverse Polish Notation (RPN) modes. You can use it to

- [Build](#), [print](#), and [plot](#) expressions containing your simulation output data
- Enter expressions, which can contain node voltages, port currents, operating points, model parameters, noise parameters, design variables, mathematical functions, or arithmetic operators, into a buffer
- Store the buffer contents into a [memory](#) and then recall the memory contents back into the buffer
- Save calculator memories to a file and load those memories back into the calculator

The help screens for the calculator assume that you are familiar with the syntax for either algebraic or RPN scientific calculators. If you have trouble entering expressions, you might want to review the manual for your own handheld scientific calculator.

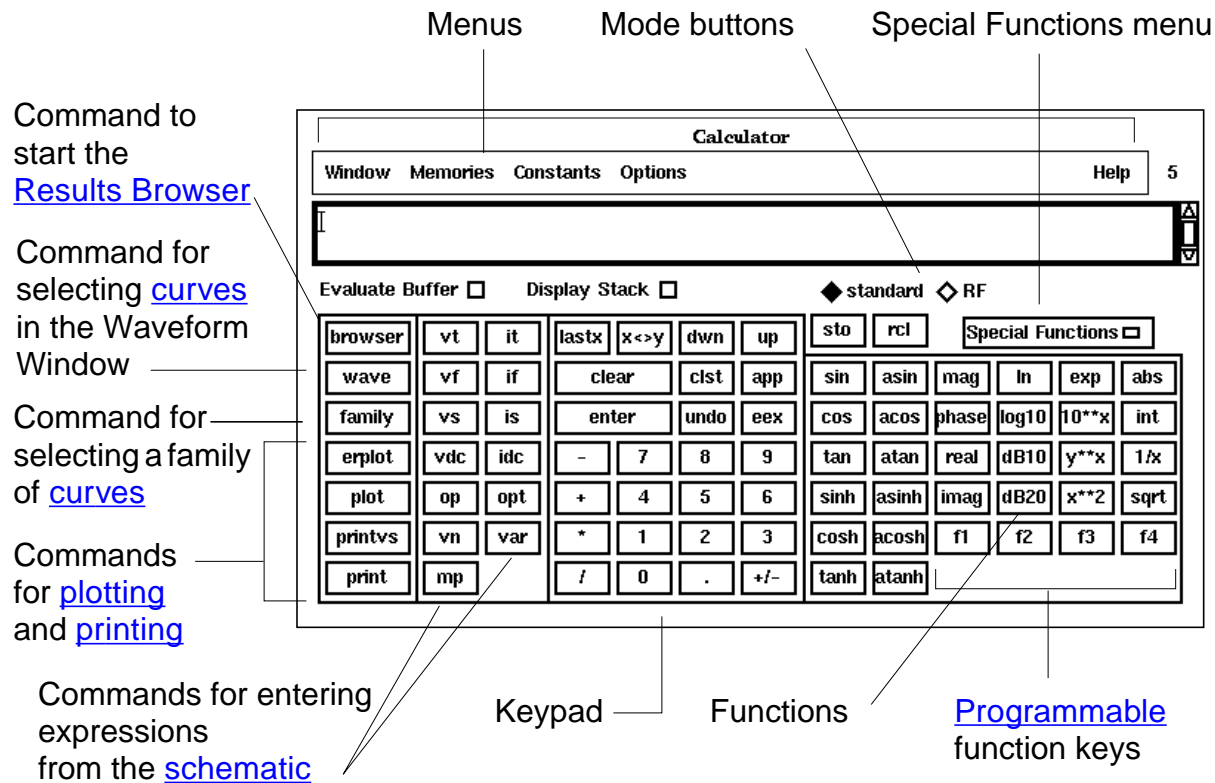
Each of the calculator functions has a corresponding SKILL command. For more information about SKILL calculator functions, refer to the [OCEAN Reference](#).

# Waveform Calculator User Guide

## Overview

## About the Calculator

The calculator has several kinds of buttons. For help on any button or area, click the labels below.



## About the Algebraic and RPN Modes

The calculator has both algebraic and Reverse Polish Notation (RPN) modes. The calculator uses the syntax rules you would expect from any scientific calculator.

To change modes

- Choose *Options – Set Algebraic*.

or

## Waveform Calculator User Guide

### Overview

---

- Choose *Options – Set RPN*.

## Using Keys

Most keys and functions are available in both modes. The following keys are unique to either the algebraic or RPN mode:

Key	Mode	Key	Mode	Key	Mode
(, ), and ()	Algebraic	lastx	RPN	app	RPN
space	Algebraic	x<>y	RPN	enter	RPN
=	Algebraic	dwn and up	RPN	clst	RPN

[Click here](#) for help using the buffer and stack in RPN mode.

## Entering Constants

To enter a constant into the buffer

- Select the constant from the Constants menu.

In Algebraic mode, the constant is added to the right side of the buffer.

In RPN mode, the current buffer expression is pushed onto the stack, and the specified constant is placed in the buffer.

The constants are pi, twoPi, sqrt2, degPerRad, charge, boltzmann, and epp0. These constants are internally defined in the Analog Expression Language (AEL) for expression evaluation. Refer to the *[Analog Expression Language Reference](#)*.

**Note:** Calculator expressions should be entered with the right syntax. In Calculator expressions, 2k, 2p etc. are interpreted as 2 kilo (2000) and 2 pico(2e-12) respectively. Expressions like '2 multiplied by p' should be entered as '2\*p' and not as '2p'. Also, any string of characters (and not integers) following say '2p', e.g. '2phfyhff' will be treated as '2p' only without any syntax errors. However, expressions like '2pghg45jk' will throw syntax errors. The same caution also applies, while using any constant like 'pi'. If the user wants to use any expression like '4 multiplied by pi', they should use '4\*pi' and not '4pi' (which will be evaluated as  $4p = 4e-12$ ).

## About Standard and RF Modes

The calculator has standard and RF modes. In RF mode, the calculator keypad provides mathematical functions commonly used in RF circuit design. This document describes both the standard and RF modes.

## Window Size and Location

You can customize the window size and location of the calculator.

- To set the default size of the calculator window, add the following command to your [~/ .cdsinit file](#):

```
armSetCalc('defaultSize width:height')
```

The width and height are in pixels (integers) and default to 490:275.

- To set the default location of the calculator window, add the following command to your [~/ .cdsinit file](#):

```
armSetCalc('defaultLocation x:y')
```

X and Y are integer screen coordinates with a default location is 0:0.

## Calculator Bindkey

You can define a bindkey for the calculator.

- Type the following in the CIW:

```
hiSetBindKey( "encap" "<Key>x" "calCalculatorFormCB()" )
```

Substitute the key you want to use for the x.

## Starting the Calculator

There are several ways to start the calculator:

- From the Waveform Window or Simulation window, choose *Tools – Calculator*.
- From the Schematic Window choose *Sim-Tools – Calculator*.
- From the CIW, choose *Tools – Mixed Signal Environment – Calculator*.
- For Analog Environment,

- ❑ From the CIW, choose *Tools - Analog Environment - Calculator*.
- ❑ From Analog Design Environment Window, choose *Tools - Calculator*.

## **Closing and Quitting the Calculator**

To close the Calculator window, but preserve the contents of the buffer and stack

- Choose *Window – Close*.

To close the Calculator window and clear the buffer and stack

- Choose *Window – Quit*.

Use this command if you need to reinitialize the calculator.

# Waveform Calculator User Guide

## Overview

---

---

## Using the Calculator

### Selecting Data

There are three ways to bring simulation results into the calculator. You can

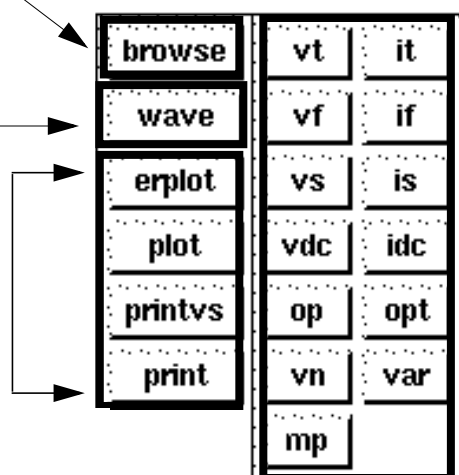
- Use the schematic expression keys to click nets and pins in the schematic and select their results
- Use the Results Browser to select results out of the UNIX file system hierarchy
- Use the *wave* command to select a curve in the Waveform Window

For more help, click the highlighted text in the figure below.

Open the Results Browser.

Bring a curve from the Waveform Window into the buffer.

Use these keys to print or plot what you select.



Schematic expression keys:  
Click a net or  
pin in the schematic.

## Selecting Data in a Schematic Window

The schematic expression keys let you enter data into the calculator buffer by selecting objects in the Schematic window.

**Note:** To use the *vn*, *var*, *op*, *opt*, or *mp* functions, you must either select results or have just run a simulation.

<i>vt</i>	transient voltage	<i>it</i>	transient current
<i>vf</i>	frequency voltage	<i>if</i>	frequency current
<i>vs</i>	source sweep voltage	<i>is</i>	source sweep current (I vs V curves)
<i>vdc</i>	DC voltage	<i>op</i>	DC operating point
<i>vn</i>	noise voltage	<i>opt</i>	transient operating point
<i>var</i>	design variable	<i>mp</i>	model parameter

1. Click a schematic expression key.
2. Click the appropriate object in the schematic.

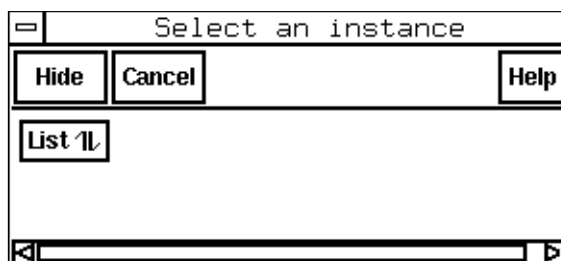
If more than one parameter is available for the expression and instance you picked, a form appears. Select the parameter you want from the [List field](#) and click *OK*.

3. When you have finished selecting objects, press the *Esc* key while the cursor is in the Schematic window.

## Choosing Parameters from Schematic Data

To select a parameter in the schematic with a [schematic expression key](#)

1. Click an instance in the schematic.

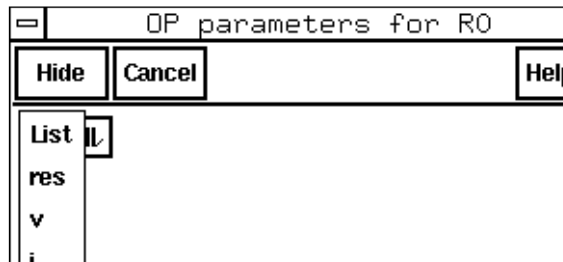


## Waveform Calculator User Guide

### Using the Calculator

---

2. Choose the parameter you want from the List field.



**Note:** When you use the *op*, *opt*, *mp*, *vn*, or *var* functions, you must have just run a simulation, or you must choose *select results* from the *Results* menu in the Simulation window. Otherwise, the system does not know what to display.

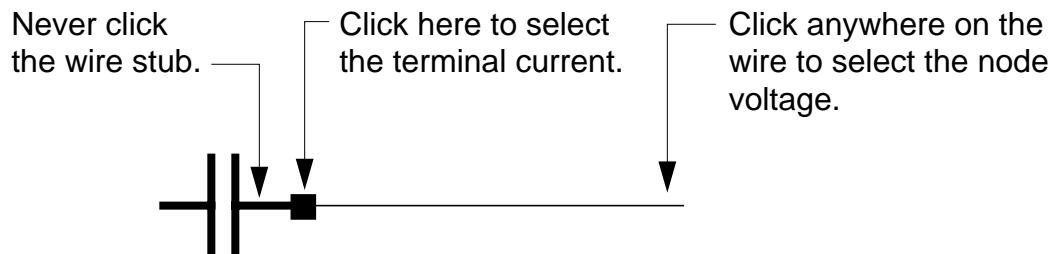
### Choosing Voltages or Currents

To select voltages in the schematic

- Click wires.

To select currents

- Click square pin symbols, not wires.



## Waveform Calculator User Guide

### Using the Calculator

---

You can use the Selection Filter form to restrict selection to either pins or wires. Press *F3* if the Selection Filter form did not appear.



The screenshot shows a dialog box titled "Selection Filter". It has a title bar with a close button (X) on the left. Below the title bar is a row of five buttons: "Hide", "Cancel", "Defaults", "Last", and "Help". The "Help" button is on the far right. Below the buttons is a list of two items, each with a checkbox to its left. The first item is "Terminal Current" with a checked checkbox. The second item is "Node Voltage" with a checked checkbox.

## Selecting Curves in the Waveform Window

Use the wave key to create an expression from a curve in the Waveform Window and place the expression into the calculator buffer.

1. Click *wave* in the calculator.
2. Click a curve in a Waveform Window.

The waveform expression that the system enters in the calculator is the expression on the Waveform Window status banner at the tracking cursor location.

If the banner expression cannot be evaluated to a waveform (because it is only a descriptive title), the system automatically creates a Cadence<sup>®</sup> SKILL language function to represent the waveform you selected.

## Selecting a Parametric Set of Curves

To select a parametric set of curves in the Waveform window

1. Click *family* in the calculator.
2. In the Waveform window, click a curved part of the parametric wave.

## Isolating One Curve from a Waveform Group

You can select a single curve from a parametric curve group in the Waveform Window. This lets you perform calculations on the single curve or display it separately on another set of axes.

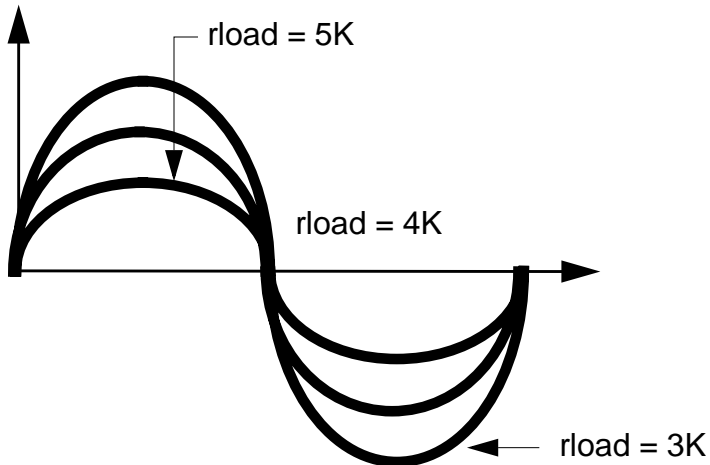
1. In the Calculator, click *wave*.

## Waveform Calculator User Guide

### Using the Calculator

---

2. Click a parametric curve in the Waveform Window.



An expression representing the individual curve appears in the calculator buffer.

3. In the Calculator, click *erplot*.

The Waveform Window is erased, and the single curve appears.

## Selecting Simulation Results with the Results Browser

To select waveform expressions from simulation output data in the Results Browser

1. Click *browse* in the calculator.

The Browse Project Hierarchy form appears.

Browse Project Hierarchy			
OK	Cancel	Defaults	Help
Project Directory		<input type="text" value="/mnt1/carolyn"/>	

2. Enter the path to the project directory in the form and click *OK*.

The Results Browser appears.

3. Click left in the Results Browser to expand the data hierarchy until you get to the scalar or waveform data you want to enter in the calculator buffer.

## Waveform Calculator User Guide

### Using the Calculator

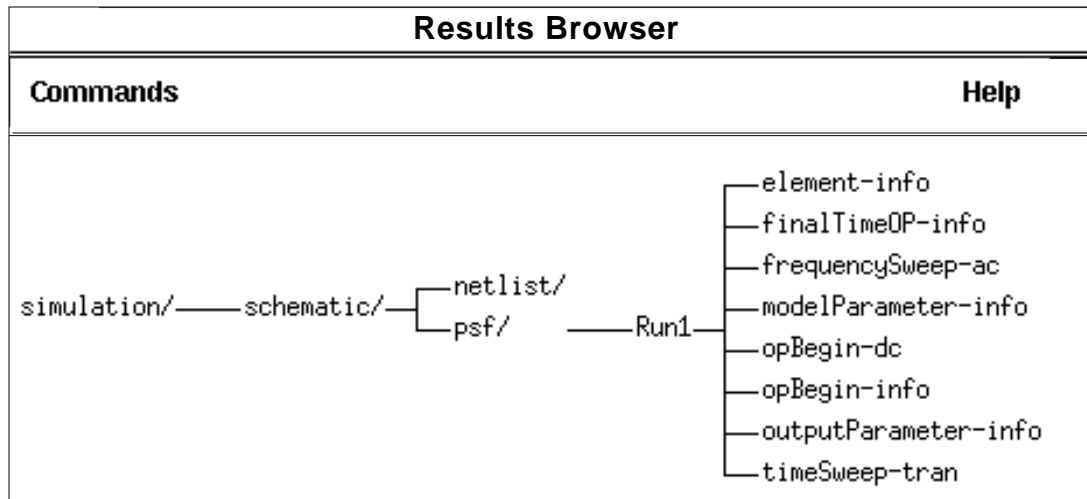
---

4. Click the scalar or waveform data to enter the expression for that data in the calculator buffer.

## Using the Results Browser

The Results (Data Results library) Browser lets you

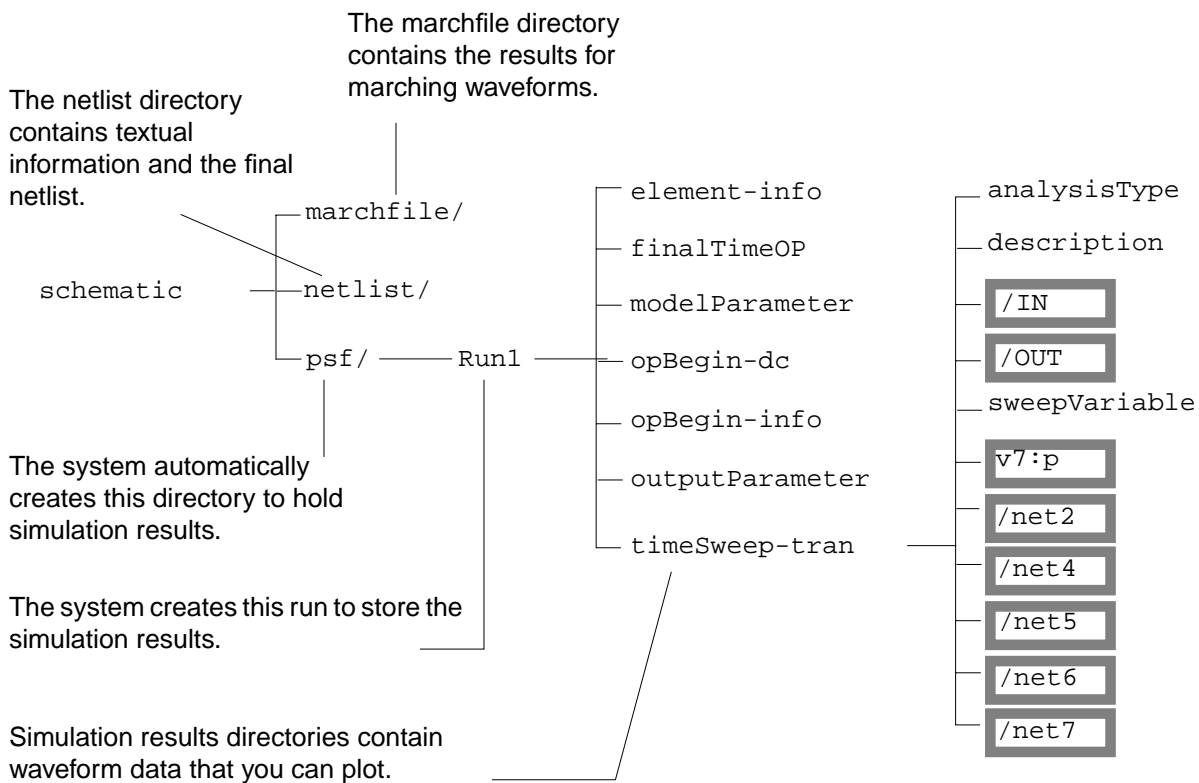
- View simulation waveforms and text results
- Plot waveforms
- Copy the waveform expression for simulation results directly into the Waveform Calculator



## Waveform Calculator User Guide

### Using the Calculator

The Results Browser stores data hierarchically in objects called [nodes](#). You can expand a particular node down to the lowest level of the hierarchy. The following figure shows the file structure for simulation results in the Results Browser.



The contents of the `psf` directory varies depending on the simulator you use. This directory might contain nodes such as the following:

Node	Contents
<code>element</code>	Component parameters for design circuit elements
<code>finalTimeOP</code>	Operating point of the component parameters at the end of the simulation
<code>modelParameter</code>	Simulation model parameters
<code>opBegin-dc</code>	Voltage of the node at $T=0$

## Waveform Calculator User Guide

### Using the Calculator

---

---

Node	Contents
<code>opBegin-info</code>	Simulator parameters at T=0
<code>outputParameter</code>	Information about the output parameters
<code>srcSweep</code>	Waveform data for the DC analysis
<code>timeSweep</code>	Waveform data for the transient analysis
<code>frequencySweep</code>	Waveform data for the AC analysis

---

Simulation results also include the following files:

---

Node	Contents
<code>analysisType</code> and <code>description</code>	Information about the type of analysis
<code>sweepVariable</code>	Information about the swept variables

---

The simulation results, which are stored in directories such as `timeSweep-tran`, might be scalar data (numeric data) or waveform data, which is highlighted.

If you press the middle mouse button over a node, a menu of commands pops up. You can use these commands to perform actions on that node. The Results Browser also has a Commands menu in the banner that helps you manipulate the display of data and lets you close the Results Browser.

## Starting the Browser

1. To start the Results Browser, do one of the following:

- ☐ Click *Browse* on the Waveform Calculator form
- ☐ Choose *Tools – Results Browser* in the Waveform Window
- ☐ Choose *Tools – Mixed Signal Environment – Results Browser* in the CIW.

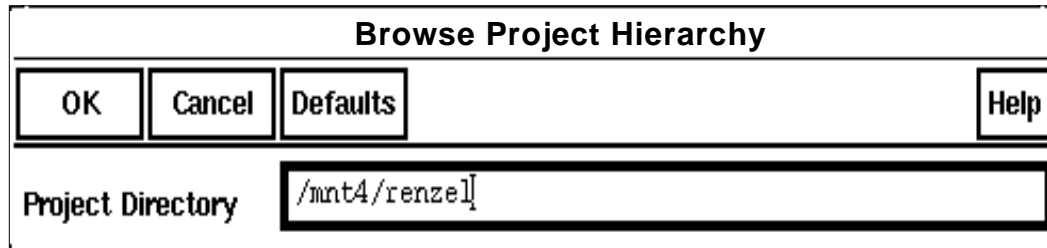
## Waveform Calculator User Guide

### Using the Calculator

---

- ❑ For Analog Environment, from the CIW, choose *Tools - Analog Environment - Results Browser*.
- ❑ For Analog Environment, from Analog Design Environment Window, choose *Tools - Results Browser*.

The Browse Project Hierarchy form appears.



**Browse Project Hierarchy**

OK Cancel Defaults Help

Project Directory /mnt4/renzel

2. Type in the path of a project directory containing simulation output files.

If you just finished a simulation, the current data directory is shown. Otherwise, the form defaults to your home directory.

3. Click *OK*.

The Results Browser appears.

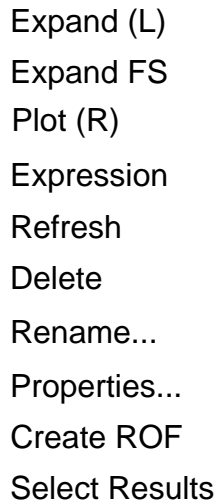
The Results Browser has a pop-up menu that lets you move through the file system and data hierarchy, perform commands on the file system and data, and display various properties of the data hierarchy.

## Waveform Calculator User Guide

### Using the Calculator

---

- To display the Results Browser pop-up menu, press the middle mouse button over a node.

A screenshot of a context menu for the Results Browser. The menu is a vertical list of options enclosed in a blue border. The options are: Expand (L), Expand FS, Plot (R), Expression, Refresh, Delete, Rename..., Properties..., Create ROF, and Select Results.

Expand (L)  
Expand FS  
Plot (R)  
Expression  
Refresh  
Delete  
Rename...  
Properties...  
Create ROF  
Select Results

← Click a command for more help.

## Managing the Display

The commands on the Results Browser menu help you manage the display of data.

To place the root (the vertical list of nodes on the left) of the Results Browser hierarchy in the middle of the window

- Choose *Commands – Root*.

After you use the scroll bar to pan through the Results Browser nodes, you can use this command to return to the default configuration of the Results Browser.

To move up a level in the Results Browser hierarchy

- Choose *Commands – Unexpand*.

To redisplay the complete Results Browser hierarchy after you add new files to the file system (with the browser open)

- Choose *Commands – Refresh*.

You can also use this command to redraw the current Results Browser display.

To limit the number of net, instance, and terminal elements displayed

1. Choose *Commands – Filter*.

## Waveform Calculator User Guide

### Using the Calculator

---

The Filter form appears.



Filter			
OK	Cancel	Defaults	Apply
Regular Expression Filter		.*	

2. Enter a regular expression filter.

This command is useful for simplifying the Results Browser display while browsing simulation results for large circuits.

To keep the most recently expanded node in the center of the window

- Choose *Commands – Centering On*.

If you do not want the most recently expanded mode to move automatically to the center of the window, you can turn off the centering mode.

- Choose *Commands – Centering Off*.

You can also use an environment variable, `browserCenterMode`, in your `.cdsenv` file to turn centering on or off. To turn centering on, add the following line to the `.cdsenv` file.

```
asimenv.misc browserCenterMode boolean t
```

To turn centering off (which is the default) add the following line to the `.cdsenv` file.

```
asimenv.misc browserCenterMode boolean nil
```

To close the Results Browser window

- Choose *Commands – Close Browser*.

### Redrawing the Browser

To redraw the Results Browser display for the current level in the Results Browser hierarchy

1. Place the pointer over the current level in the Results Browser.
2. Press the middle mouse button and choose *Refresh* from the pop-up menu.

## Waveform Calculator User Guide

### Using the Calculator

---

**Note:** To refresh the entire Results Browser hierarchy, choose *Commands – Refresh* on the Results Browser menu.

### Configuring the Browser

You can customize the size and location of the Results Browser window.

- To set the default size of the Results Browser window, add the following line to your `~/cdsinit` file:

```
armSetBrowser('defaultSize width:height')
```

The width and height are in pixels (integers), and they default to 800:500.

- To set the default location of the Results Browser window, add the following line to your `~/cdsinit` file:

```
armSetBrowser('defaultLocation x:y')
```

x and y are the screen coordinates (integers) of the location you want, and they default to 0:0.

**Note:** To customize the Results Browser for the current session only, enter these commands on the input line of the CIW.

### Expanding Nodes

In the Results Browser, a node represents an object that contains information. Scalar data (numeric data) and waveform objects are highlighted.

- To expand a node, click left on it.
  - ☐ If you click a file node, the file contents appear in a file viewing window. You can use the window functions to manipulate the file.
  - ☐ If you click a directory node, the directory contents appear in the next level of the Results Browser hierarchy. If the directory contains simulation data, the data files open and the simulation data hierarchy expands.
  - ☐ If you click a highlighted object, the scalar data or the expression for the waveform object is entered into the [calculator buffer](#) for mathematical processing.

**Note:** You can also expand a node by choosing the *Expand (L)* command from the Results Browser pop-up menu.

To see an example of an expanded hierarchy of nodes containing simulation results, [click here](#).

## Waveform Calculator User Guide

### Using the Calculator

---

You can use the *Expand FS* command from the Results Browser pop-up menu to expand only the UNIX file system—not the [psf hierarchy](#) that the system uses to display simulation results.

To expand the UNIX file system for a node

- Press the middle mouse button over the node and choose *Expand FS*.

The UNIX file system expands to show any directories or files under the node.

## Plotting Waveforms

You can plot several waveforms from the same or different simulations in the [Waveform Window](#).

To plot a waveform object in the Waveform Window

1. Expand the Results Browser nodes until you see highlighted waveform data.

Waveform data is in the [psf directory](#), which is under the simulation results for the simulator you chose.

2. Click right over the waveform data.

The waveform data is plotted in the Waveform Window. When you plot a waveform this way, it is added to the existing waveforms in the Waveform Window.

To erase the existing waveforms and plot only the selected waveform, use the *Window – Reset* command in the Waveform Window before you click right on the node to plot.

**Note:** You can also use the *Plot (R)* command from the Results Browser pop-up menu to plot waveform objects in the Waveform Window.

## Copying Expressions into the Calculator

To copy the expression for a Results Browser object to the calculator buffer

- Click left over a highlighted object in the Results Browser.

The expression for the object appears in the calculator buffer.

**Note:** You can also use the *Expression* command from the Results Browser pop-up menu to enter expressions in the calculator buffer.

You can use this command to enter expressions for

## Waveform Calculator User Guide

### Using the Calculator

---

- Operating points
- Model parameters
- Noise parameters
- Waveform data objects
- Scalar data

### Deleting UNIX Files

To delete a file or directory from the UNIX file system

1. Place the pointer over the file or directory you want to delete.
2. Press the middle mouse button and choose the *Delete* command from the pop-up menu.

A dialog box asks if you want to delete the file or directory.

A warning message appears in the CIW if the node you select is not part of the file system.

3. Click *OK* on the dialog box.

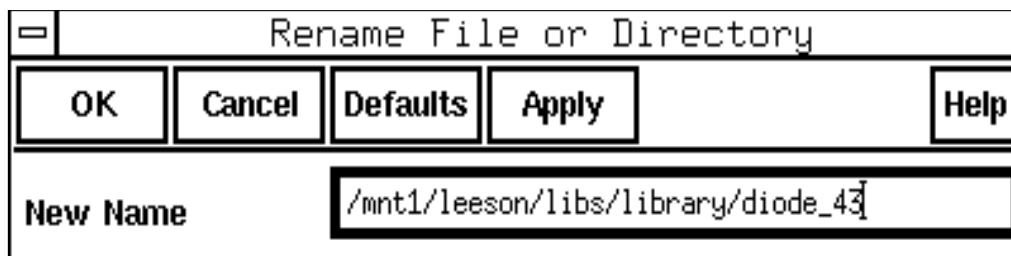
The directory or file is deleted from the file system.

### Renaming UNIX Files

To rename a file or directory in the UNIX file system

1. Place the pointer over the file or directory you want to rename.
2. Press the middle mouse button and choose the *Rename* command from the pop-up menu.

The Rename File or Directory form appears.



Rename File or Directory				
OK	Cancel	Defaults	Apply	Help
New Name	/mnt1/leeson/libs/library/diode_43			

3. Enter the new file or directory name and click *OK*.

## Displaying Properties

You can view the properties associated with the simulation output data hierarchy and filter the properties so that only the simulation properties for a particular type of component appear. (A property is an associated value, such as a DC operating point.)

1. Place the pointer over a simulation output data object, which is any node under the *psf* directory.

Simulation data is in the [psf directory](#), which is under the simulation results for the simulator you chose.

2. Press the middle mouse button and choose the *Properties* command from the pop-up menu.

The Properties Filter form appears.

3. Choose a Matching Type.

**Matching Type** lets you choose the type of pattern matching.

- ☐ **csh** uses the same pattern-matching tools used by the UNIX C-shell to match file names. This is the default.
- ☐ **regex** uses the standard UNIX regular expression pattern- matching syntax.

4. Change the filter string.

The default filter string, which is an asterisk (\*), matches everything. You can change the default to match any string you want. For example, to match all the resistors in a schematic with the *csh* matching mechanism, set the filter string to *R\**.

5. Click *OK* on the form.

The results appear in a text window.

## Using External Data

You can use the *Create ROF* command to create a run object file, which lets you access external data generated by a standalone simulator. (When you use the Analog Artist simulation environment to run simulations, this file is generated automatically for you.)

To create a run object file

1. Place the pointer over a directory that contains the external simulation data.
2. Press the middle mouse button and choose the *Create ROF* command from the pop-up menu.

A run object file called runObjFile is created in the next level of the hierarchy.

3. Click left on the node containing the simulation results to expand it.

**Note:** You cannot see the run object file unless you use the *Expand FS* command to expand the file system.

## Selecting Results

You can link data from a previous simulation to a schematic window. Then you can probe the schematic to compare the results of the previous simulation to current results.

1. Place the pointer over a node containing simulation results.
2. Press the middle mouse button and choose *Select Results* from the pop-up menu.

The schematic window is redrawn and the CIW tells you that the simulation results for the node are selected.

**Note:** You can also use the *Results – Select Results command* from the Simulation window.

## Plotting or Printing Results

You can plot or print the value of the calculator buffer expression against an independent variable.

You can plot or print only the expressions that are in the buffer, not the memories. You must recall memory expressions into the buffer to plot or print them.

You can print or plot multiple expressions by separating each expression by a space using the *app* (append) function in RPN mode or the space key in algebraic mode. For example,

## Waveform Calculator User Guide

### Using the Calculator

---

`expr1 expr2 expr3`

## Plotting Expressions

To erase the Waveform Window and plot the buffer expression

- Click *erplot* in the calculator.

To plot the buffer expression without first erasing the Waveform Window

- Click *plot* in the calculator.

For example, to plot the I vs. V curve after a DC source-sweep analysis

1. In the calculator, click *I**S*.
2. In the schematic, click the output terminal of the device.

Terminals are the square symbols at the end of the wire stub.

Now you have an expression in the buffer for the IV curve.

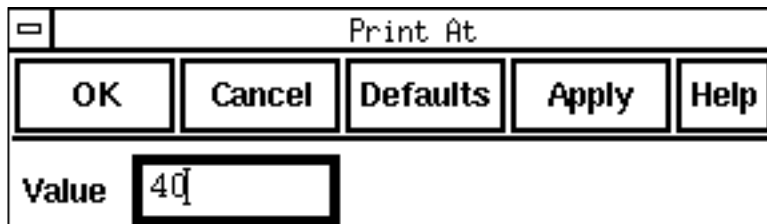
3. Click *erplot* in the calculator.

The system opens a Waveform Window (unless one is already open) and draws the curve.

## Printing One Expression Value

The *print* command prints the value of an expression at a single value of the independent variable.

1. Put the expression into the calculator buffer.
2. Click *print*.
3. If the expression is a waveform, enter the value of the independent variable.



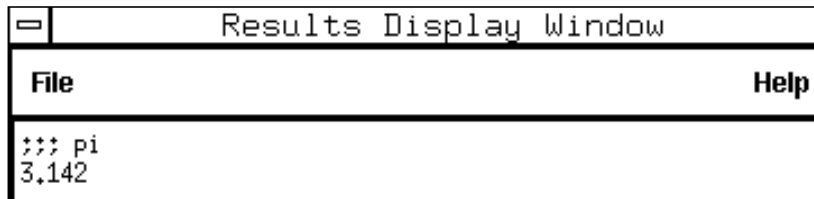
Print At	
OK	Cancel Defaults Apply Help
Value	40

## Waveform Calculator User Guide

### Using the Calculator

---

The Results Display Window appears, displaying the results in numerical format.



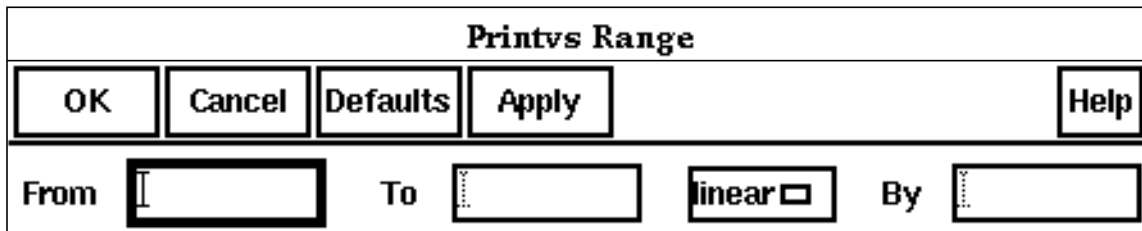
**Note:** If the expression in the calculator buffer represents a parametric waveform, then the *print* button prints the value of the expression at the specified point for different sets of sweep parameters.

### Printing a Range of Expression Values

The *printvs* command prints a table showing the value of the buffer expression over a range of independent variables.

1. Put the expression into the calculator buffer.
2. Click *printvs*.

The Printvs Range form appears.



3. Type in the starting and ending values.

Leave all of the fields blank to print the raw simulation data.

4. (Optional) Choose a linear or logarithmic range.
5. Enter the [increment](#) for the calculation in the By field.

**Note:** When results related to signals represent multi-bit buses, the *printvs* button prints the data corresponding to each bit. For example, if you use Calculator to print `VDC ( " /x<2:4>" )`, where `x<2:4>` is a sub-bus of the bus `x<0:5>` and `x<2>=3`, `x<3>=3.5`, and `x<4>=4`, you would get:

```
bit      VDC ( "x<2:4>" )
```

## Waveform Calculator User Guide

### Using the Calculator

---

0	3
1	3.5
2	4

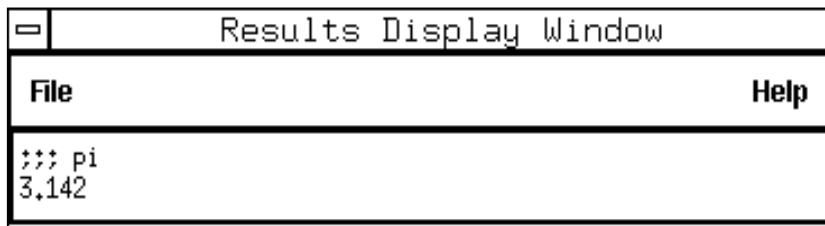
For the bus represented as  $x<2:4>$ , the swept parameter *bit* varies from 0 to the length of the bus/sub-bus, which is 2 in this example. So, bit0 is  $x<2>$ , bit1 is  $x<3>$ , and bit4 is  $x<4>$ .

### Setting the Range Calculation Increment

You can set the range calculation increment in the Printvs Range form in several ways:

- If the range is linear, enter the increment.
- If the range is logarithmic, enter the number of points per decade.
- To print the raw simulation data, leave all three fields blank.

The result of the evaluation is displayed in the Results Display Window.

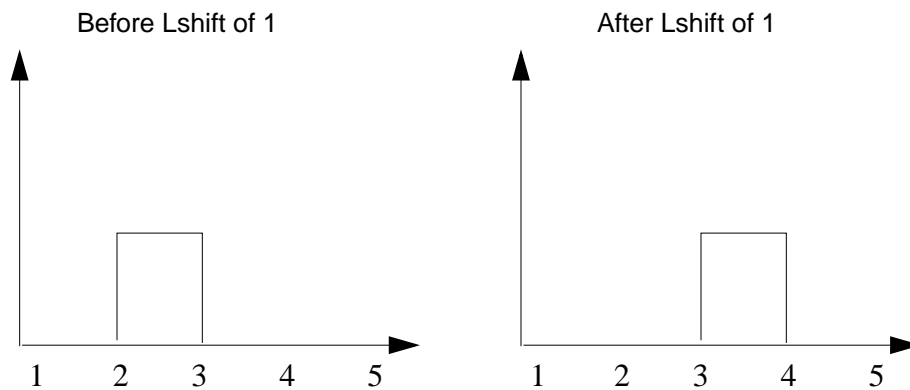


**Note:** If you specify range values outside the range of the original simulation data, the calculator extends the simulation endpoint values to the requested values using linear extrapolation, which can give misleading results.

## Applying Waveform Transformations

### Shifting the X Axis (L shift)

The *Lshift* function shifts the X axis in the Waveform Window to the left by a specified amount. Use negative values to shift the X axis to the right.



### In Algebraic Mode

In algebraic mode you set up the waveform expression after selecting the *Lshift* function.

1. From the Special Functions menu, choose *Lshift*.

The Left Shift form appears.

Left Shift	
OK	Cancel Defaults Apply Help
Get Buffer	<input type="text"/>
Delta X	<input type="text" value="0"/>

2. Enter a value in X axis units in the Delta X field, and click *OK*.

## Waveform Calculator User Guide

### Using the Calculator

---

#### In RPN Mode

In RPN mode the *Lshift* function acts on the expression already in the buffer:

1. From the Special Functions menu, choose *Lshift*.

The Left Shift form appears.

Left Shift				
OK	Cancel	Defaults	Apply	Help
Delta X	<input type="text" value="0"/>			

2. Enter a value in X axis units in the Delta X field and click *OK*.

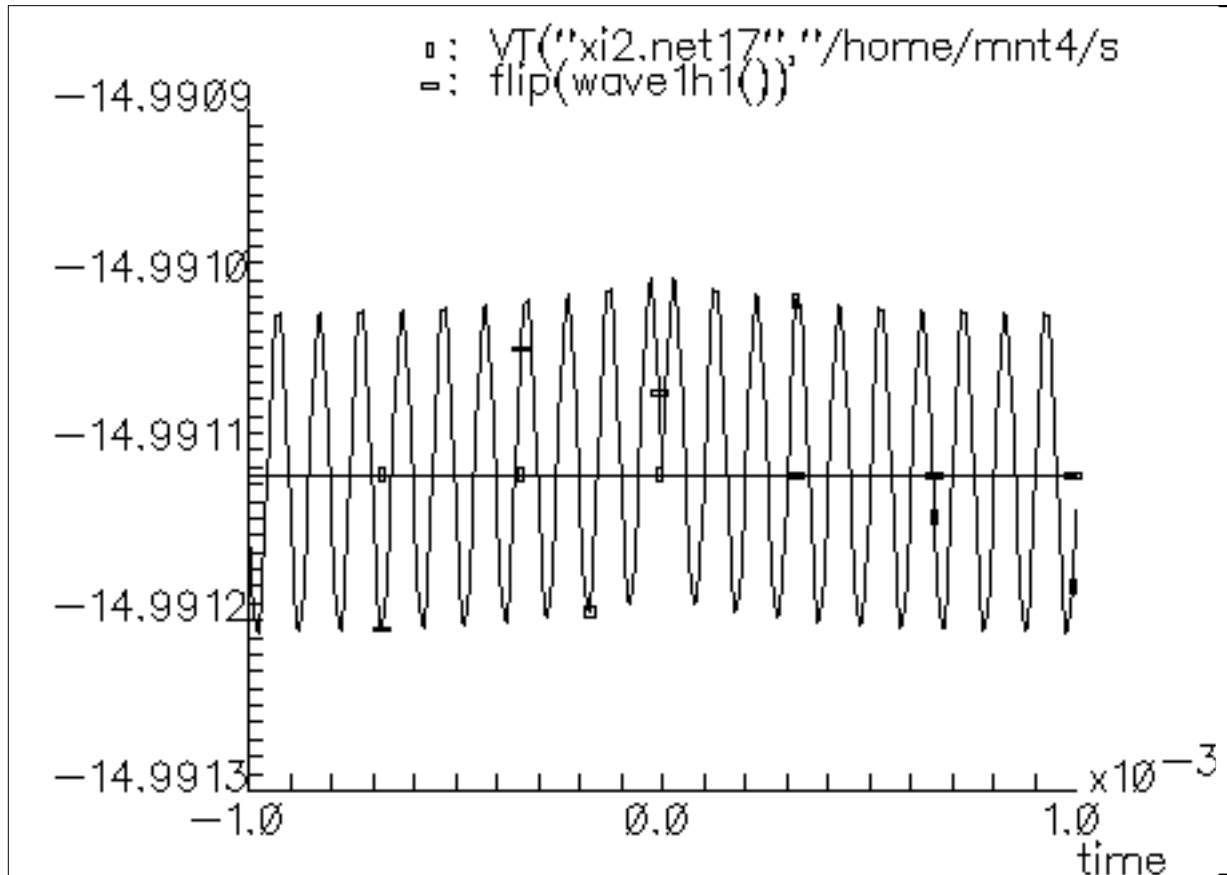
#### Y Axis Flip

To flip a waveform around the y axis

## Waveform Calculator User Guide

### Using the Calculator

- From the Special Functions menu, choose *flip*.



## Using Memories

You can store the buffer in a memory and recall it later. You can also save the calculator memories to a file. Use the Memories menu to work with memories.

### Storing Memories

To store the current buffer expression in a memory

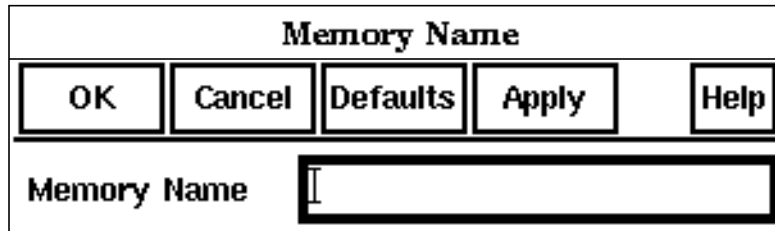
1. In the calculator, choose *Memories – Store* or click *sto*.

## Waveform Calculator User Guide

### Using the Calculator

---

The Memory Name form appears.



The Memory Name dialog box has a title bar labeled "Memory Name". Below the title bar is a row of five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons is a label "Memory Name" followed by a text input field with a cursor inside.

2. Type a name for the memory and click *OK*.

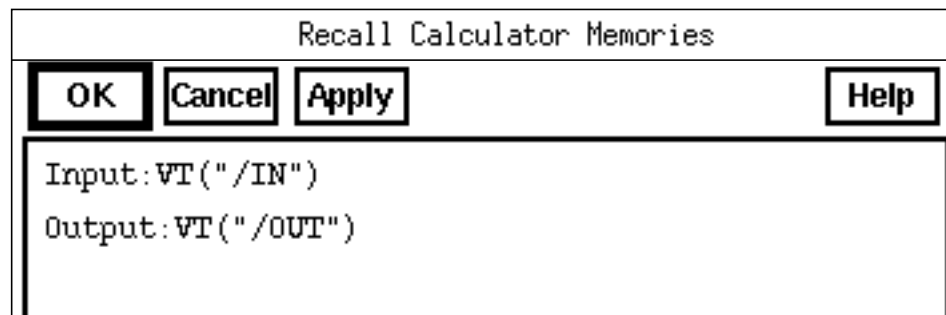
There is no limit on the number of memories you can use.

## Recalling Memories

To recall a memory into the buffer

1. Choose *Memories – Recall* or click *rcl*.

The Recall Calculator Memories form appears.



The Recall Calculator Memories dialog box has a title bar labeled "Recall Calculator Memories". Below the title bar is a row of four buttons: "OK", "Cancel", "Apply", and "Help". Below the buttons is a text area containing the following text:  
Input: VT("/IN")  
Output: VT("/OUT")

2. Double click to select an expression, or highlight the expression and click *OK*.

The recalled expression stays in the calculator memory pool.

In Algebraic mode, the recalled expression is appended to the end of the buffer. The memory *asfmem* stores the buffer temporarily while you enter a special function expression.

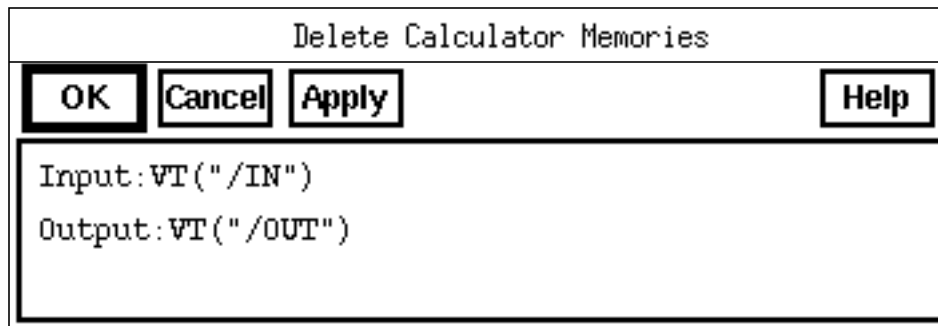
In RPN mode when you recall a memory, any expression currently in the buffer is pushed onto the calculator stack.

## Deleting Memories

To delete an expression from the memory

1. Click *Memories – Delete*.

The Delete Calculator Memories form appears.



The dialog box is titled "Delete Calculator Memories". It has four buttons at the top: "OK", "Cancel", "Apply", and "Help". Below the buttons is a text area containing the following text:

```
Input: VT("/IN")
Output: VT("/OUT")
```

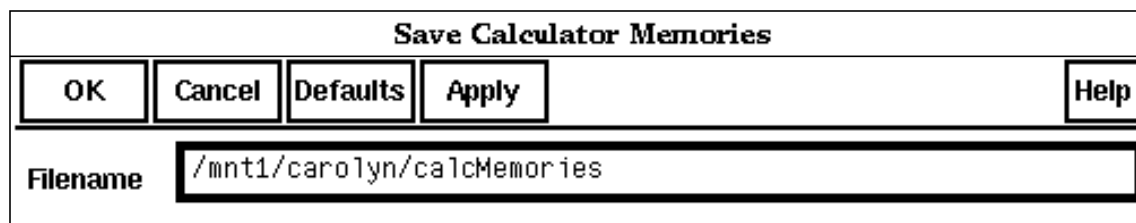
2. Double click to delete an expression, or highlight the expression and click *OK*.

## Saving and Loading Memories

Use the *Save* command to write memories to a file. Use the *Load* command read memories from a saved file.

1. Choose *Memories – Save* or *Memories – Load*.

The Save Calculator Memories form or the Load Calculator Memories form appears.



The dialog box is titled "Save Calculator Memories". It has five buttons at the top: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons is a text field labeled "Filename" with the following text:

```
/mnt1/carolyn/calcMemories
```

2. Enter the path and name for the memory file.
3. Click *OK*.

## Defining Functions and Function Keys

### Defining New Functions

You can define a function and add it to the Special Functions menu with the following steps.

1. Define the [form](#) that prompts for user-defined arguments to the special function.
2. Define the [syntax](#) of the special function in the callback procedure.
3. [Register](#) the special function.

### Defining a Form

The following example shows how to define an input form for a function that takes three arguments. The first argument is the buffer expression. The other two arguments are the boundaries of the range of the expression on which you want to operate.

```
procedure( CreateMyForm()  
  let( ( fieldList a b )  
    a = ahiCreateStringField(  
      ?name 'from  
      ?prompt "From"  
      ?value ""  
    )  
    b = ahiCreateStringField(  
      ?name 'to  
      ?prompt "To"  
      ?value ""  
    )  
    fieldList = list(  
      list( a 5:0 120:25 40 )  
      list( b 160:0 110:25 30 )  
    )  
    calCreateSpecialFunctionsForm( 'MyForm  
      fieldList )))
```

In this example, the From and To fields are string fields created in a two-dimensional form specification for fieldList. The form is created by the call to [calCreateSpecialFunctionsForm](#). This function creates and registers the form with the specified form symbol, MyForm.

## Waveform Calculator User Guide

### Using the Calculator

---

#### Defining a Callback Procedure

You define a callback procedure that is called from the entry on the Calculator Special Functions menu. Since this example uses a form to prompt for additional information required by the special function, the callback procedure is

```
procedure( MySpecialFunctionCB()  
  calCreateSpecialFunction(  
    ?formSym 'MyForm  
    ?formInitProc 'CreateMyForm  
    ?formTitle "Test"  
    ?formCallback "calSpecialFunctionInput( 'test  
      '(from to) )"  
  )  
)
```

In this procedure, a call is made to [calCreateSpecialFunction](#), which creates and displays the form and then builds the expression in the buffer with the specified form fields.

#### Using Stack Registers in the Procedure

You can use the special symbol 'STACK in the list of form fields to get expressions from the stack.

For example, if you want to insert a stack element between the From and To arguments in the special function expression, you could specify the callback line as follows:

```
?formCallback "calSpecialFunctionInput('test '(from STACK to))"
```

If your special function does not require a form to prompt for additional arguments, you can define your callback as follows:

```
procedure( MySpecialFunctionCB()  
  calSpecialFunctionInput( 'test nil )  
)
```

#### Registering the Function

You register the function and callback with [calRegisterSpecialFunction](#):

```
calRegisterSpecialFunction(  
  list( "test" 'MySpecialFunctionCB )  
)
```

The next time you select *Calculator* from the Waveform or Simulation window menu, your new special function appears in the Special Functions menu.

## **SKILL User Interface Functions for the Calculator**

For SKILL Functions of Waveform Calculator, please refer to *chapter 22* of [Cadence Analog Design Environment SKILL Language Reference](#).

## **Assigning Function Keys**

You can assign buffer and stack manipulation procedures to the four function keys *f1*, *f2*, *f3*, and *f4*. To do this, use SKILL commands that you type in the CIW or add to your [.cdsinit file](#).

For example, you can use the *f1* key to create the expression for the magnitude and phase of an AC waveform in the buffer by defining the following RPN mode procedure:

```
procedure(f1( )
    calCalcInput('(enter phase xchxy mag append))
)
```

This [calCalcInput](#) function manipulates the buffer containing the expression

```
VF("/net1")
```

to create the expression

```
mag(VF("/net1")) phase(VF("/net1"))
```

# **Waveform Calculator User Guide**

## Using the Calculator

---

---

## RPN Mode

This section describes how you use the calculator in RPN mode.

### About the Buffer

The buffer area stores data and expressions that you enter for the calculator to manipulate.

You can enter expressions into the buffer in several ways:

- Use the keypad and function keys.
- Read calculator memories.
- Type at the keyboard.

You can edit the buffer contents by backspacing to delete unwanted characters.

**Note:** If you enter multiple expressions, separate them with one or more spaces.

### Using the Keypad, Functions, and Memories

You can select numbers, functions, operators, and constants from the keypad. The calculator automatically adds the selection to the buffer.

The functions on the keypad (such as log) make the entire buffer's contents the argument of the functions. For example, if the buffer contains `expr1` and you select log, the following appears in the buffer:

```
log(expr1)
```

Select a function when there is only one expression in the buffer. If the buffer contains multiple expressions, the space between expressions causes an error, as in the following example:

```
log(expr1 expr2)
```

To read a calculator memory into the buffer, choose [Memories – Recall](#).

## Entering Variables

To enter a variable expression into the buffer

1. Select the *var* button from the Waveform Calculator form.

The Select an Instance form appears.

2. On the schematic, choose an instance that uses this variable as a parameter value.
3. Choose the variable from the *List* cyclic button on the Select an Instance form.
4. Select *OK*.

The expression appears in the buffer.

**Note:** To use the *var* function, you must either select results or have just run a simulation.

## Entering Constants

To enter a constant into the buffer

- Select the constant from the Constants menu.

The current buffer expression is pushed onto the stack, and the specified constant is placed in the buffer.

The constants are pi, twoPi, sqrt2, degPerRad, charge, boltzmann, and epp0. These constants are internally defined in the Analog Expression Language (AEL) for expression evaluation. Refer to the [\*Analog Expression Language Reference\*](#).

## Entering Multiple Expressions

You can enter more than one expression into the buffer with the app (append) key. Each expression must be separated by a space, which you can enter from the keyboard.

Many functions cannot operate on multiple expressions. Be careful not to enter one of these functions while you have more than one expression in the buffer.

You can build the expressions separately in the stack and then combine them in the buffer with the app (append) key.

## Evaluating the Buffer

Evaluating the buffer is useful only for expressions that contain scalar functions or variables. The Waveform Calculator evaluates expressions that contain waveforms or undefined variables as NaN (not a number).

To evaluate an expression in the buffer after the arithmetic operations are performed on it

- Select the Evaluate Buffer option.

## About the Stack

Like a Reverse Polish Notation (RPN) scientific calculator, the Waveform Calculator uses a [buffer](#) and a stack to build and manipulate expressions. One-expression functions operate on the buffer only and leave the contents of the stack unaffected. Multiple-expression functions operate on both the buffer and the stack elements.

An RPN calculator performs arithmetic operations by positioning the expressions in the stack the same way you would on paper. For example, to add  $a$  and  $b$ , you write down the two numbers and then perform the addition. Similarly, you always position the expressions for an operation in the calculator in the natural order first, then enter the arithmetic operator.

The stack performs many movements automatically to help you perform long chain calculations.

The stack “lifts” every expression in the stack when a new expression is entered and “drops” expressions into position when you perform multiple-expression operations.

Because operations are performed when the function is executed, the length of the expression being built is limited only by the Cadence® SKILL language string length (maximum 8191 characters). In addition, the stack drops during calculations involving the buffer and the first stack register. This lifting and dropping of the stack lets you retain intermediate expressions in long calculations without having to store the expressions into other memory locations.

By starting every problem at its innermost set of parentheses and working outward (just as you would with pencil and paper) you maximize the efficiency and power of the RPN calculator logic.

## Displaying the Stack

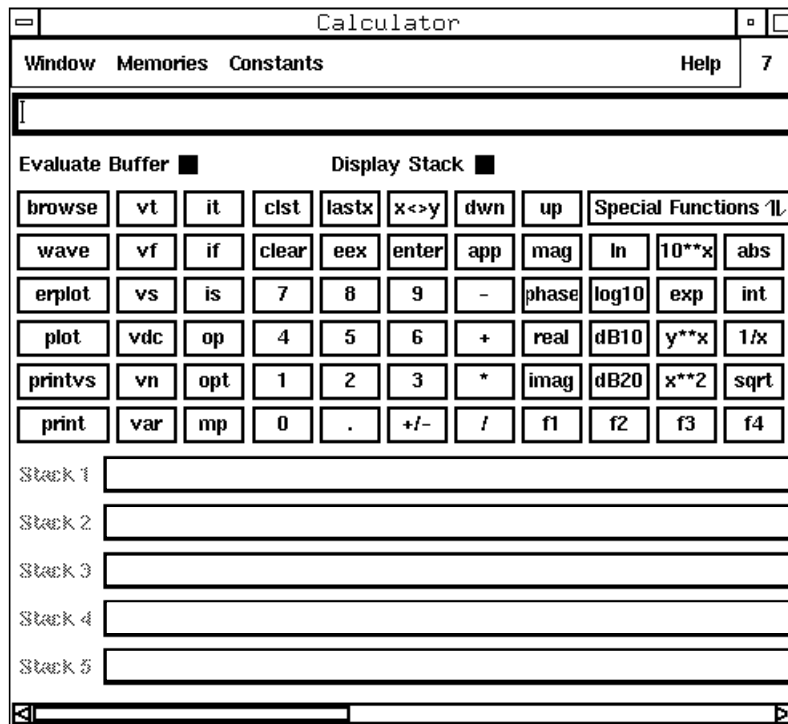
To display the top five elements in the stack

## Waveform Calculator User Guide

### RPN Mode

- Select the Display Stack option.

You can use this option to help you understand how the Waveform Calculator performs mathematical operations. It lets you look at the first five registers of the stack simultaneously so you can see the effect of a given keypad sequence.



## Moving the Buffer and Stack Register

You can move expressions within the stack, and between the stack and [buffer](#), with the *enter*, *dwn*, *up*, and *x< >y* functions.

To recall the previous expression into the buffer, use the *lastx* key.

## Pushing Expressions on the Stack

When you make an entry into the calculator buffer, you must tell the calculator that you have finished and are ready to make the next entry.

To push the current expression onto the stack

- Click *enter*.

Unless the buffer is empty, *enter* duplicates the contents of the buffer and pushes it onto the stack. All the existing expressions in the stack are pushed down one register. For efficiency, many calculator functions perform an implicit *enter* operation.

## Exchanging the Buffer and Stack

Use the up and dwn commands to exchange the contents of the buffer and the stack registers.

To move the contents of the first stack register into the buffer and move the contents of the buffer to the bottom of the stack

1. Select *dwn*.
2. Keep selecting *dwn* to cycle the buffer and stack to their original states.

To scroll the stack in the opposite direction

- Select *up*.

To exchange the contents of the buffer and the first stack register without changing the contents of the remaining stack registers

- Select *x< >y* (x exchange y)

## Recalling the Last Expression

The lastx register contains the last expression in the buffer upon which a mathematical operation was performed.

To place the contents of the lastx register into the buffer

- Select *lastx*.

The calculator pushes the current buffer expression onto the stack.

The lastx register is most useful in calculations where an expression is used more than once. By recovering an expression using lastx, you avoid manually reentering that expression into the calculator.

For example, to enter the expression  $(1+x)/x$ , use the following key sequence (with the evaluate buffer option turned off):

```
1  
enter
```

```
clear  
x  
+  
lastx  
/
```

## Clearing the Buffer and Stack

There are several ways to clear the calculator buffer and stack.

To remove a single character from the buffer

- Press the backspace or delete key.

To clear the buffer without affecting the stack

- Click *clear* on the calculator.

To clear the buffer and stack

- Click *c/st* on the calculator.

## Operators and Functions

The calculator has both algebraic and Reverse Polish Notation (RPN) modes.

**Note:** Each of the calculator functions has a corresponding SKILL command. For more information about SKILL calculator functions, refer to the [OCEAN Reference](#).

In RPN mode the calculator uses the syntax rules you would expect from any handheld scientific calculator.

For example, to enter the function  $(1+x)/x$  in RPN mode, you would use this key sequence

```
1 enter clear x + lastx /
```

Help is also available for

- The [buffer](#) and [stack](#)
- Algebraic mode

## Waveform Calculator User Guide

### RPN Mode

### Single-Expression Functions

These functions operate on only a single expression in the buffer.

Key	Function	Key	Function
<i>mag</i>	magnitude	<i>exp</i>	$e^x$
<i>phase</i>	phase	$10^{**}x$	$10^x$
<i>real</i>	real component	$y^{**}x$	$y^x$
<i>imag</i>	imaginary component	$x^{**}2$	$x^2$
<i>ln</i>	base-e (natural) logarithm	<i>abs</i>	$ x $ (absolute value)
<i>log10</i>	base-10 logarithm	<i>int</i>	integer value
<i>dB10</i>	dB magnitude for a power expression	$1/x$	inverse
<i>dB20</i>	dB magnitude for a voltage or current	<i>sqrt</i>	$\sqrt{x}$

**Note:** Selecting these functions while the buffer contains multiple expressions is an error in RPN mode because there is a space between expressions. For example, `ln(expr1 expr2)` is invalid because the logarithm function takes only one argument, not two.

### Example: Plotting the Magnitude of a Signal

To plot the dB magnitude of a signal after an AC analysis in RPN mode

1. Click *vf* on the calculator.
2. On the schematic, click the net you want to plot.
3. With the cursor in the Schematic window, press the *Esc* key.  
This cancels the *vf* function. Otherwise, the command stays active.
4. Click *dB20* on the calculator.  
The calculator buffer now contains the expression you want to plot.
5. Click *plot* to show the curve.

## Two-Expression Functions and Operators

In RPN mode, two-expression functions operate on both the buffer and the first stack element.

Key	Function in RPN Mode
$y^{**}x$	$y^x$ (in RPN mode, evaluated as $\text{stack}_1^{\text{buffer}}$ )
<i>app</i>	Appends the first stack element to the end of the buffer expression, separated by a space. This operation lets you display multiple waveform expressions in RPN mode.  <b>Note:</b> This key is not in algebraic mode.
+	Adds the buffer expression to the first stack register.
-	Subtracts the buffer expression from the first stack register.
*	Multiplies the buffer expression by the first stack register.
/	Divides the first stack register by the buffer expression.

### Example: Instantaneous Power Dissipation

This example computes the instantaneous power dissipated by a resistor.

1. Click *vt*.
2. On the schematic, click the net connected to the appropriate pin of the resistor.
3. With the cursor in the Schematic window, press the *Esc* key.  
This cancels the *vt* function. Otherwise, the command stays active.
4. Click *it*.
5. Click the appropriate pin of the resistor.  
To select currents, click the square pin symbol. Do not click the wire stub.
6. Click \* on the calculator.
7. Click *plot*.

## Trigonometric Functions

The trigonometric functions work like the other single-expression functions. Selecting these functions while the buffer contains multiple expressions creates an error in RPN mode because there is a space between expressions.

sin	asin
cos	acos
tan	atan
sinh	asinh
cosh	acosh
tanh	atanh

## Special Functions

Special functions help you analyze waveform data generated with calculator expressions. Some functions pop up a form where you enter the data required for the calculation. Other special functions act directly on the data currently in the buffer (RPN mode).

<i>xmax</i>	<i>fourEval</i>	<i>psdbb</i>
<i>xmin</i>	<i>frequency</i>	<i>riseTime</i>
<i>ymax</i>	<i>gainBwProd</i>	<i>rms</i>
<i>ymin</i>	<i>gainMargin</i>	<i>rmsNoise</i>
<i>average</i>	<i>groupDelay</i>	<i>root</i>
<i>bandwidth</i>	<i>harmonic</i>	<i>sample</i>
<i>clip</i>	<i>harmonicFreq</i>	<i>settlingTime</i>
<i>compression</i>	<i>iinteg</i>	<i>slewRate</i>
<i>compressionVRI</i>	<i>integ</i>	<i>spectralPower</i>
<i>convolve</i>	<i>ipn</i>	<i>stddev</i>
<i>cross</i>	<i>ipnVRI</i>	<i>table</i>

## Waveform Calculator User Guide

### RPN Mode

---

<i>dBm</i>	<i>lshift</i>	<i>tangent</i>
<i>delay</i>	<i>overshoot</i>	<i>thd</i>
<i>deriv</i>	<i>phaseMargin</i>	<i>value</i>
<i>dft</i>	<i>phaseNoise</i>	<i>xval</i>
<i>flip</i>	<i>psd</i>	

---

### Example: Average Value of a Current

This example shows how to compute the average value of a current during the simulation period.

1. Click *it* on the calculator.
2. On the schematic, click the terminal whose current you want to average.  
To select currents, click the square pin symbol. Do not click the wire stub.
3. With the cursor in the Schematic window, press the *Esc* key.  
This cancels the *it* function. Otherwise, the command stays active.
4. Choose *Special Functions – Average*.
5. Click *print* on the calculator.

The system displays the average value in a text window.

## RPN Mode Special Functions

### Average Function

The *average* function computes the average of a waveform over its entire range. Average is defined as the integral of the expression  $f(x)$  over the range of  $x$ , divided by the range of  $x$ . For example, if  $y=f(x)$ ,  $average(y) =$

$$\frac{\int_{from}^{to} f(x)dx}{to - from}$$

where *to* and *from* are the range of *x*.

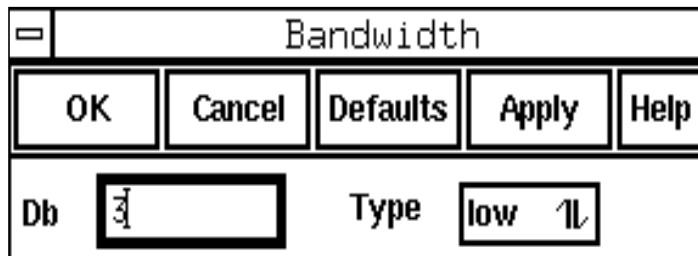
If you want a different range, use the [clip function](#) to clip the waveform to the range you want.

## Bandwidth Function

The *bandwidth* function calculates the bandwidth of the waveform in the calculator buffer. Please note that the input waveform must represent a true voltage, NOT modified by a dB.

1. Select *bandwidth*.

The Bandwidth form appears.



Bandwidth				
OK	Cancel	Defaults	Apply	Help
Db	3	Type	low	⌵

2. In the *Db* field, enter how far below the peak value you want to see data.
3. Choose a bandwidth response from the *Type* field.
  - ☐ [low](#) computes the bandwidth of a low-pass response.
  - ☐ [high](#) computes the bandwidth of a high-pass response.
  - ☐ [band](#) computes the bandwidth of a band-pass response.
4. Click *OK*.

## Computing Low-Pass Bandwidth

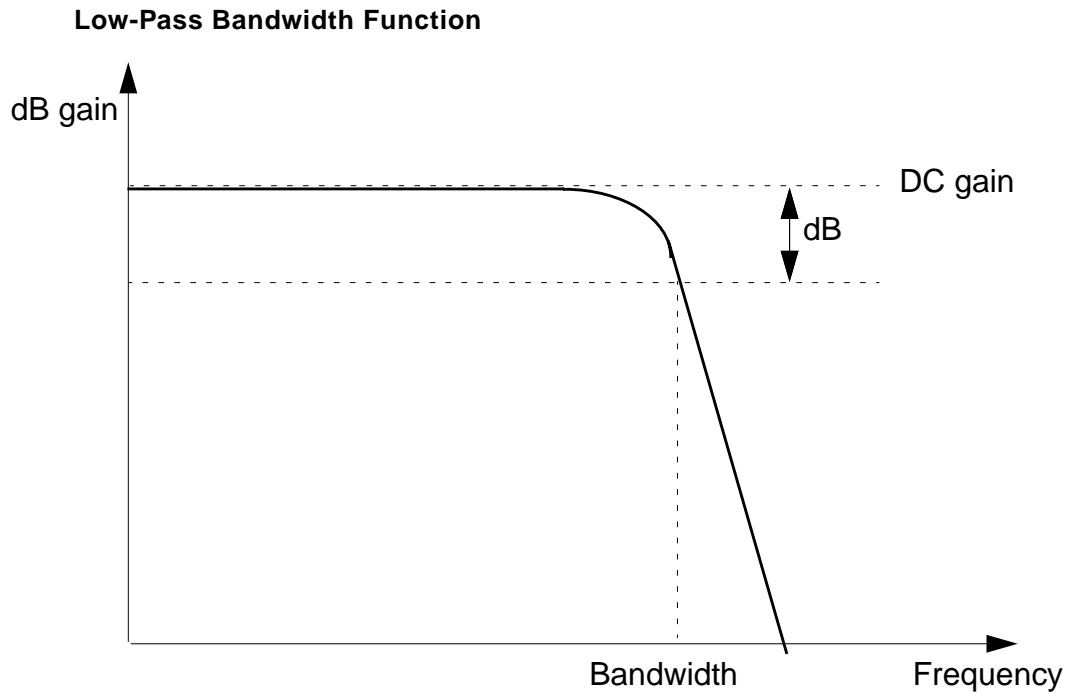
The calculator computes the low-pass bandwidth by determining the smallest frequency at which the magnitude of the input waveform drops *n* decibels below the DC gain. (DC gain is obtained by zero-order extrapolation from the lowest or highest computed frequency, if

## Waveform Calculator User Guide

### RPN Mode

---

necessary.) The *dB* field specifies *n*. An error occurs if the magnitude of the input waveform does not drop *n* decibels below the DC gain.



### Computing High-Pass Bandwidth

The calculator computes the high-pass bandwidth by determining the largest frequency at which the magnitude of the input waveform drops *n* decibels below the gain at the highest

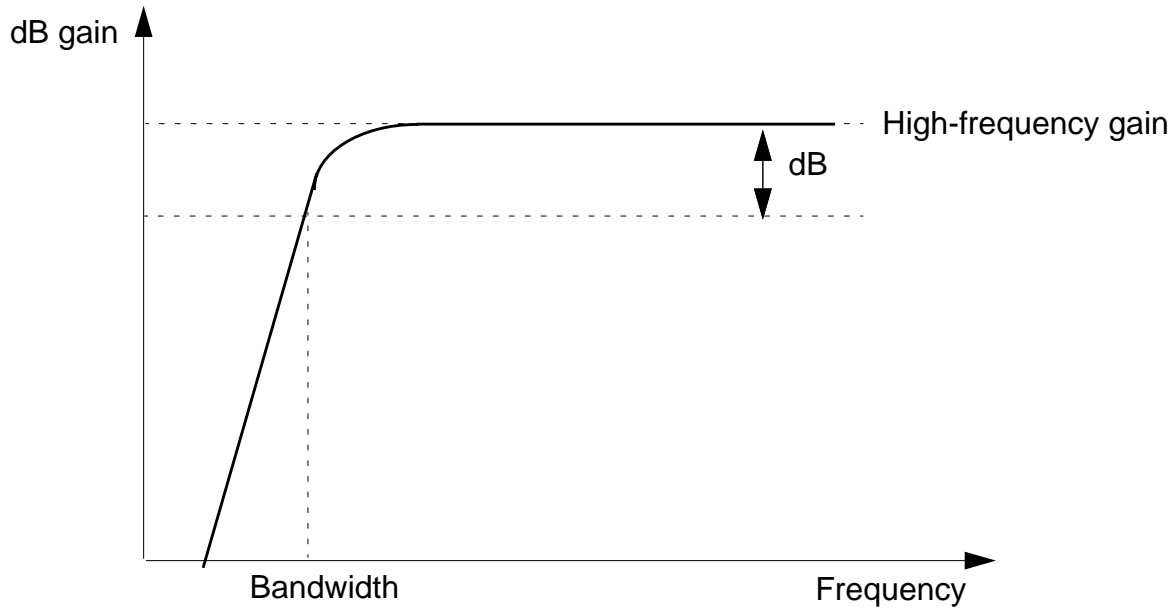
## Waveform Calculator User Guide

### RPN Mode

---

frequency in the response waveform. The *dB* field specifies *n*. An error occurs if the magnitude of the input waveform does not drop *n* decibels below the gain at high frequency.

#### High-Pass Bandwidth Function



#### Computing Band-Pass Bandwidth

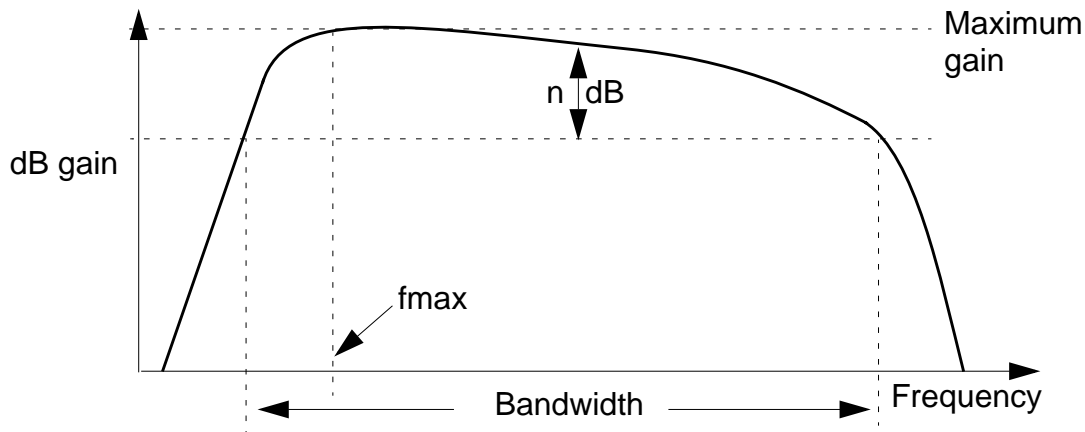
The calculator computes the band-pass bandwidth by

1. Determining the lowest frequency ( $f_{\max}$ ) at which the magnitude of the input waveform is maximized
2. Determining the highest frequency less than  $f_{\max}$  at which the input waveform magnitude drops *n* decibels below the maximum (*n* is the number you enter in the *dB* field)
3. Determining the lowest frequency greater than  $f_{\max}$  at which the input waveform magnitude drops *n* decibels below the maximum

## Waveform Calculator User Guide

### RPN Mode

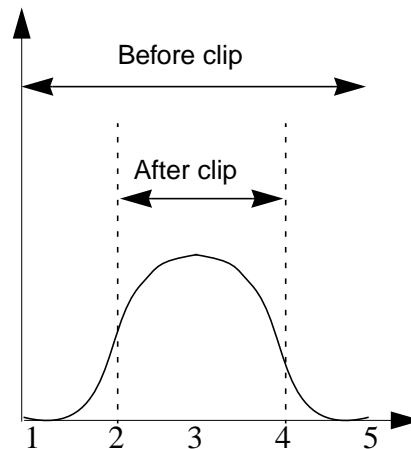
- Subtracting the value returned by step 2 from the value returned by step 3. The value returned by step 2 or step 3 must exist.



## Clip Function

The *clip* function restricts the waveform defined by the buffer expression to the range entered in the *From* and *To* fields. You can use the *clip* function to restrict the range of action of other special functions of the calculator such as [integ](#), [rms](#), and [frequency](#).

Clip				
OK	Cancel	Defaults	Apply	Help
From	<input type="text" value="2"/>	To	<input type="text" value="4"/>	



## Compression Function

This function returns the *Nth* compression point value of a waveform at the extrapolation point that you specify. To use this function:

## Waveform Calculator User Guide

### RPN Mode

---

1. Set up the ne600p mixer cell from `dfII/samples/artist/rfExamples` library. The design variable `frf` should be set to 920MHz.
2. Ensure that the sourcetype on the rf port is 'sine' and the Amplitude (dBm) is set to `prf`.
3. Set up a PSS analysis. The beat frequency should be set to 40MHz. Set the number of harmonics to 2 (only two harmonics are required to determine the 1 dB compression point). Sweep the `prf` parameter from -30 to 10 in 10 linear steps.
4. Set the Model Library path to include the `dfII/samples/artist/models/spectre/rfModels.scs` file.

5. After running the simulation, call up the Waveform Calculator and the Results Browser. Click on *schematic->psf->Run1->sweep\_pss\_fd-sweep->sweepVariable->prf->10->sweep\_pss-004\_pss-fd.pss->Pif* with the left mouse button. The following will appear in the calculator buffer:

```
v( "/Pif" ?result "sweep_pss_fd-sweep" ?resultsDir "~/simulation/ne600p/spectre/schematic" ).
```

6. Click on *Special Functions -> compression*. The *Compression* form will be displayed.

Compression			
OK	Cancel	Defaults	Apply
Help			
Harmonic Num.	<input type="text" value="2"/>	Ext. Point (X)	<input type="text" value="0"/>
Compression dB	<input type="text" value="1"/>		

7. Enter *Harmonic number*=2. This is the second harmonic of the 40 MHz fundamental frequency, which is the IF frequency (80MHz).
8. Enter *Ext. Point (X)* = -25 field to specify the extrapolation point of the waveform. The extrapolation point is the X axis value.
9. Enter *Compression dB* = 1 to specify the compression coefficient (N).
10. Click on the *OK* button.
11. Click the *Evaluate Buffer* button in the Calculator. The result appears in the Calculator display.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## CompressionVRI Function

This function performs an Nth compression point measurement on a power waveform.

Use this function to simplify the declaration of a compression measurement. This function extracts the specified harmonic from the input waveform(s), and uses  $\text{dBm}(\text{spectralPower}((i \text{ or } v/r), v))$  to calculate a power waveform. The function then passes this power curve and the remaining arguments to the *compression* function to complete the measurement.

The *compression* function uses the power waveform to extrapolate a line of constant slope (dB/dB) according to a specified input or output power level. This line represents constant small-signal power gain (ideal gain). The function then finds the point where the power waveform drops N dB from the constant slope line and returns either the x coordinate (input referred) or y coordinate (output referred) value.

To use this function:

1. Define the voltage waveform in the buffer.
2. Choose *compressionVRI* in the *Special Functions* menu.

The CompressionVRI form opens.

The screenshot shows a dialog box titled "CompressionVRI". At the top, there are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons, there are four input fields arranged in a 2x2 grid. The top-left field is labeled "Harmonic", the top-right is "Extrapolation Point", the bottom-left is "Load Resistance", and the bottom-right is "Compression dB". Each field has a small "Default" label next to it. At the bottom of the dialog, there is a checkbox labeled "Input Referred Compression" which is currently checked.

3. Type a value in the *Harmonic* field to specify the harmonic index of the waveform.
4. Type a value in the *Extrapolation Point* field to specify the extrapolation point for the waveform. The default value is the minimum x value of the input voltage waveform.

The extrapolation point is the coordinate value in dBm that indicates the point on the output power waveform where the constant-slope power line begins. This point should be in the linear region of operation.

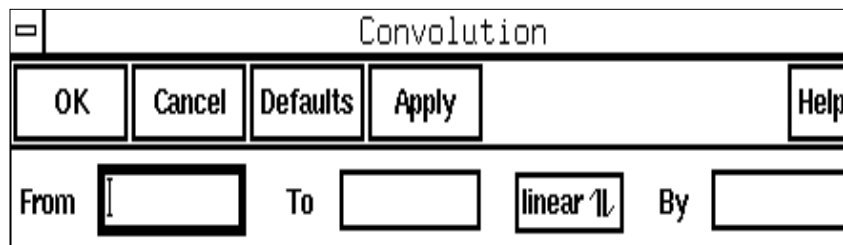
5. Type a numerical value in the *Load Resistance* field. The default value is 50.
6. In the *Compression dB* field, type the delta (in dB) between the power waveform and the ideal gain line that marks the compression point. The default value is 1.
7. Choose whether the measurement is for *Input Referred Compression* or *Output Referred Compression*.
8. Click *OK*.

## Convolution (convolve) Function

The *convolve* function computes the convolution of two waveforms.

1. Define the first waveform in the buffer.
2. Define the second waveform as the first stack element.
3. Choose *convolve* from the *Special Functions* menu.

The Convolution form appears.



Convolution is defined as

$$\int_{\text{from}}^{\text{to}} f1(s)f2(t-s)ds$$

f1 and f2 are the functions defined by the first and second waveforms.

**Note:** The *convolve* function is numerically intensive and might take longer than the other functions to compute.

## Threshold Crossing (cross) Function

The *cross* function computes the x-axis value *xcross* at which the *n*th crossing of the specified edge type of the threshold value occurs.

1. Choose *cross*.

The Threshold Crossing form appears.

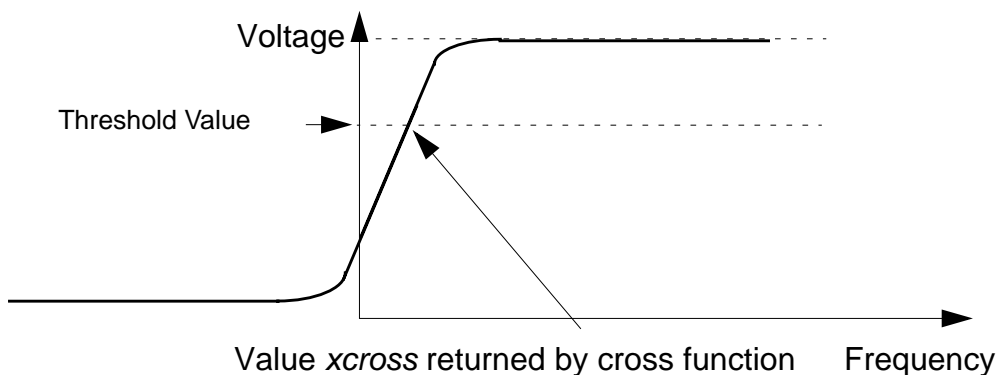
Threshold Crossing					
OK	Cancel	Defaults	Apply	Help	
Threshold Value	<input type="text" value="4.5"/>	Edge Number	<input type="text" value="1"/>	Edge Type	<input type="text" value="rising"/>

2. Enter the threshold value of the waveform at which to perform the calculation.
3. In *Edge Number*, enter the number of the crossing at which to perform the calculation.

The integer you enter specifies which crossing is returned. (For example, 1 specifies the first crossing, and 2 specifies the second crossing.)

If you specify a positive integer, the count starts at the smallest *x* value of the waveform, and the search is in the direction of increasing *x* values. If you specify a negative integer, the count starts at the largest *x* value of the waveform, and the search is in the direction of decreasing *x* values. If you enter 0, all the crossings found are returned in a list.

4. Select an *Edge Type* to determine the crossing as the rising edge, falling edge, or either edge.



5. Click *OK*.

## dBm Function

The *dBm* function performs the operation

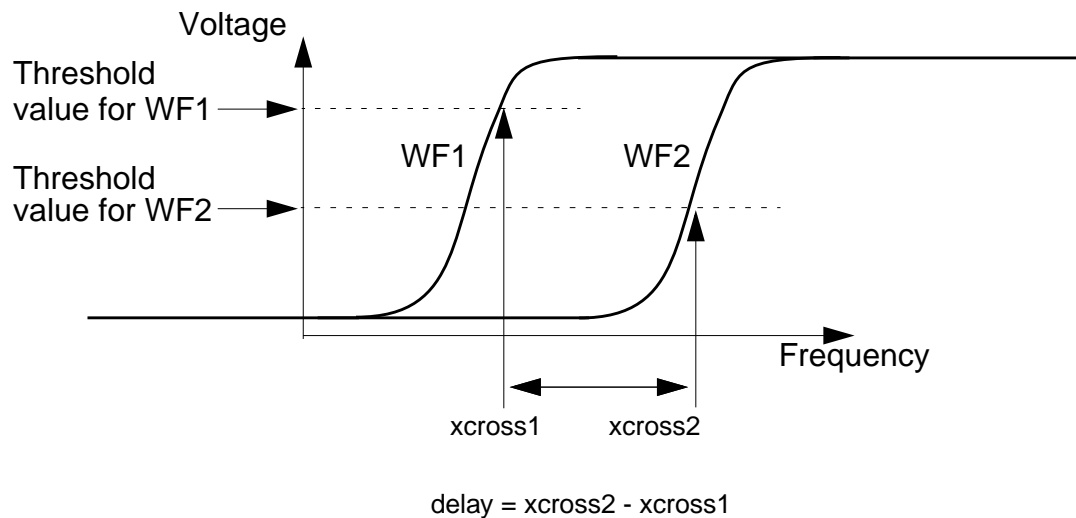
$$\text{dB10}(x) + 30$$

1. Enter the value in the buffer.
2. Select *dBm*.

## Delay Function

The *delay* function computes the delay between two points using the [cross function](#).

1. Enter the second waveform into the first stack register.
2. Enter the first waveform into the buffer.
3. Select *delay* and fill in the Threshold Delay form.



Threshold Delay			
OK	Cancel	Defaults	Apply
<a href="#">Help</a>			
Wf1: Threshold Value	<input style="width: 50px;" type="text" value="2&lt;sup&gt;5&lt;/sup&gt;"/>	Edge Number	<input style="width: 50px;" type="text" value="1"/>
Edge Type	<input style="width: 100px;" type="text" value="either 1"/>		
Wf2: Threshold Value	<input style="width: 50px;" type="text" value="2&lt;sup&gt;5&lt;/sup&gt;"/>	Edge Number	<input style="width: 50px;" type="text" value="1"/>
Edge Type	<input style="width: 100px;" type="text" value="either 1"/>		

## Derivative (deriv) Function

The *deriv* function computes the derivative of the buffer expression. You can plot the resulting waveform.

1. Select *deriv*.
2. Enter the expression and closing parenthesis into the calculator buffer.

**Note:** After the second derivative, the results become inaccurate because the derivative is obtained numerically.

## Discrete Fourier Transform (dft) Function

The tool which converts a temporal (time domain) description of a signal (real or complex) into one, in terms of its frequency components is called the Fourier transform. DFT (Discrete Fourier Transform) is the discrete formulation of the Fourier transform, which takes such regularly spaced data values (samples in time domain), and returns the value of the Fourier transform for a set of values in frequency domain which are equally spaced. Most of the time, however, we work on real-valued signals only.

Consider a complex series (signal)  $w(k)$  with  $N$  samples of the form

$$w(0), w(1), w(2), \dots, w(k), \dots, w(N-1)$$

Further, assume that the series outside the range  $0, N-1$  is extended  $N$ -periodic, that is,  $w(k) = w(k+N)$  for all  $k$ . The DFT of this series will be denoted  $W(n)$ , will also have  $N$  samples and will be defined as:

$$W(n) = \frac{1}{N} \sum_{k=0}^{N-1} w(k) \left( e^{-2\pi i k \frac{n}{N}} \right) \quad \text{where } n = 0, \dots, N-1$$

## Waveform Calculator User Guide

### RPN Mode

---

#### Note:

- The first sample  $W(0)$  of the transformed series is the DC component, more commonly known as the average of the input series.
- The DFT of a real series results in a symmetric series about the Nyquist frequency (described below).
- The highest positive (or negative) frequency sample is called the Nyquist frequency. This is the highest frequency component that should exist in the input series for the DFT to receive 'unpredictable' results. More specifically, if there are no frequencies above Nyquist frequency, the original signal can be exactly reconstructed from the samples. The Nyquist Theorem (or Shannon's Sampling Theorem) exactly specifies this, that for a band limited signal, you must sample at a frequency over twice the maximum frequency of the signal, to reconstruct it from the samples.

While the DFT transform above can be applied to any complex valued series, in practice for large series it can take considerable time to compute, the time taken being proportional to the square of the number of points (samples) in the series. A much faster algorithm has been developed by Cooley and Tukey called the FFT (Fast Fourier Transform). The only requirement of the most popular implementation of this algorithm (Radix-2 Cooley-Tukey) is that the number of points in the series be a power of 2 i.e.  $N=2^n$ .

Given  $N$  input points, the FFT returns  $N$  frequency components, of which the first  $(N/2 + 1)$  are valid. (The other components are mirror images and are considered invalid since the frequencies they represent do not satisfy the Nyquist Theorem above.) They start with the DC component, and are spaced apart by a frequency of  $(1 / (n \text{ } \delta T))$ . The magnitude of the complex number returned is the frequency's relative strength.

The *dft* function computes the discrete Fourier transform of the buffer by FFT algorithm where  $\delta T = (t_2 - t_1) / N$ . The waveform is sampled at the following  $N$  timepoints:

$t_1, t_1 + \delta T, t_1 + 2 * \delta T, \dots, t_1 + (N - 1) * \delta T$

The output of *dft()* is a frequency waveform,  $W(f)$ , which has  $(N/2 + 1)$  complex values: the dc term, the fundamental, and  $(N/2 - 1)$  harmonics.

**Note:** The last time point,  $(t_1 + (N - 1) * \delta T)$ , is  $(t_2 - \delta T)$  rather than  $t_2$ . The *dft* function assumes that  $w(t_1)$  equals  $w(t_2)$ . To use the *dft* function

1. Select *dft*.

## Waveform Calculator User Guide

### RPN Mode

---

The Discrete Fourier Transform form is displayed.

The screenshot shows a dialog box titled "Discrete Fourier Transform". It has a standard Windows-style title bar with a minimize button. Below the title bar are four buttons: "OK", "Cancel", "Defaults", and "Apply". In the top right corner is a "Help" button. The main area of the dialog contains several input fields and labels: "From" and "To" are text boxes for specifying a range; "Number of Samples" is a text box with the value "64"; "Window Type" is a dropdown menu currently showing "Rectangular"; "Smoothing Factor" is a text box with the value "1"; and "Coherent Gain" is a dropdown menu currently showing "(none)".

2. Specify the range over which you want to compute the transform.

Be sure to cover at least one complete period of your slowest frequency.

3. Enter the *Number of Samples* you want to take in expanding the Fourier transform.

This number should be a power of 2. If it is not, the system increases the value to the next higher power of 2. Sample at a rate that is at least twice your highest frequency component (the Nyquist rate). Pick a sampling rate high enough that closely spaced frequency components can be resolved.

4. Select the *Window Type* option.

For more information, see the [table](#) of window type option values later in this section.

5. Specify the *Smoothing Factor* (for the Kaiser window type only).

The *Smoothing Factor* field accepts values from 0 to 15. The value 0 implies no smoothing and is equivalent to a rectangular window. The default value for the *Smoothing Factor* field is 1.

6. Click *OK*.

When you run the transient analysis, keep the maximum time step small enough to represent the highest frequency component accurately. The maximum time step should be smaller than the sampling period that you use for the discrete Fourier transform (DFT) of the time domain waveform. The samples in the DFT will either hit a data point (calculated exactly by the simulator) or an interpolated point between two data points.

Choosing a maximum timestep during transient simulation that is smaller than the DFT sampling period ensures that sampling maintains a resolution at least equal to that of the transient time-domain waveform.

## Waveform Calculator User Guide

### RPN Mode

The start and stop times should not coincide with the boundaries of the time-domain waveform. The boundary solutions might be imprecise and generate incorrect results if used in other calculations.

One of the uses of fast Fourier transform (FFT) windowing is to reduce discontinuities at window edges caused by having a nonintegral number of periods of a signal in a window. This removes the abrupt edges, making them fall off smoothly to zero, and can improve the validity of the FFT components obtained. You can also use FFT windowing to 'dig out' the details of signal components that are very close in frequency or that consist of both large and small amplitudes.

The following table was obtained from the book *The FFT, Fundamentals and Concepts* by R. W. Ramirez, Prentice Hall, 1985. As explained in this reference, the values in the table were computed from software-generated windows and might vary slightly from theoretical values. In the third column, the peak magnitude of each window is compared with that of the rectangular window. In the fourth column, the amplitude of the highest side lobe is given in decibels referenced to the major lobe peak. The fifth column contains the 3dB bandwidth of the major lobe, normalized to one over the window's width. The last column gives the theoretical rolloff of the side lobes.

Window name	Shape equation	Major lobe height	Highest side lobe (dB)	Band-width (3 dB)	Theor. rolloff (dB/octave)
Cosine4	$A = (.5(1 - \cos(2\pi t/T)))^2$ for $t=0$ to $T$	$0.36T$	-46.9	$1.79/T$	30
ExtCosBell	$A = 0.5(1 - \cos(2\pi 5t/T))$ for $t=0$ to $T/10$ and $t=9T/10$ to $T$ $A=1$ for $t=T/10$ to $9T/10$	$0.9T$	-13.5	$0.95/T$	18 (beyond $5/T$ )
HalfCycleSine	$A = \sin(2\pi 0.5t/T)$ for $t=0$ to $T$	$0.64T$	-22.4	$1.15/T$	12
HalfCycleSine3	$A = \sin^3(2\pi 0.5t/T)$ for $t=0$ to $T$	$0.42T$	-39.5	$1.61/T$	24
HalfCycleSine6					
Hamming	$A = .08 + .46(1 - \cos(2\pi t/T))$ for $t = 0$ to $T$	$0.54T$	-41.9	$1.26/T$	6 (beyond $5/T$ )
Kaiser					

## Waveform Calculator User Guide

### RPN Mode

Window name	Shape equation	Major lobe height	Highest side lobe (dB)	Bandwidth (3 dB)	Theor. rolloff (dB/octave)
Parzen	$A = 1 - 6\left(\frac{2t}{T} - 1\right)^2 + 6\left \frac{2t}{T} - 1\right ^3$ for $t = T/4$ to $3T/4$ $A = 2(1 -  2t/T - 1 )^3$ for $t = 0$ to $T/4$ and $t = 3T/4$ to $T$	$0.37T$	-53.2	$1.81/T$	24
Rectangular	$A = 1$ for $t = 0$ to $T$	$T$	-13.2	$0.86/T$	6

### Sources of Errors

- *dft()* performs interpolation to determine values of  $w(t)$  that are not directly available from the simulator output. This interpolation can cause an inaccurate spectrum.
- If  $(t_2 - t_1)$  is not the time period of  $w(t)$ , the output of *dft()* might be misleading.
- If the simulator generated values are inaccurate, *dft()* returns a frequency waveform with many 'insignificant' harmonics. You can minimize the amplitudes of these harmonics by increasing the accuracy of the simulator.

### eyeDiagram Function

The *eyeDiagram Function* gives an eye-diagram plot in which the waveform signal is divided into fixed time periods, which are then superimposed on each other. The result is a plot that has many overlapping lines enclosing an empty space known as the "eye". The quality of the receiver circuit is characterized by the dimension of the eye. An open eye means that the detector will be able to distinguish between 1's and 0's in its input, while a closed eye means that a detector placed on  $V_{out}$  is likely to give errors for certain input bit sequences.

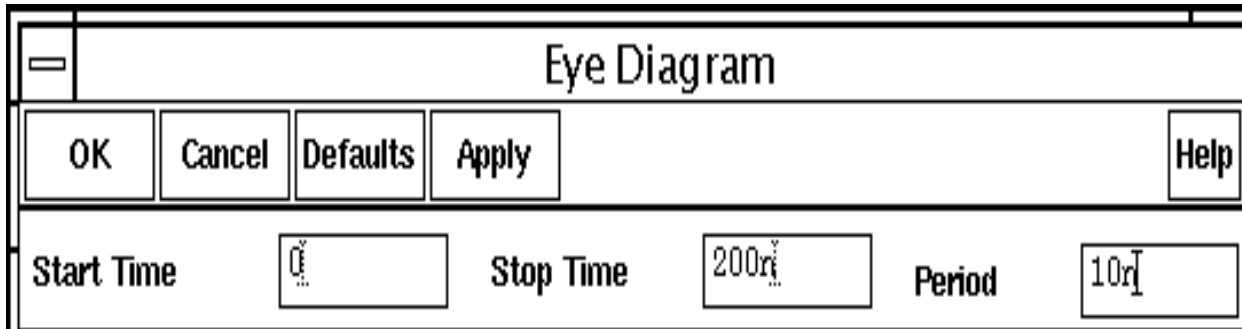
1. Define a waveform in the buffer.

## Waveform Calculator User Guide

### RPN Mode

---

2. Choose *eyeDiagram* from the *Special Functions* menu in *Calculator*. The Eye Diagram form appears.



The screenshot shows a dialog box titled "Eye Diagram". It has a standard window control button (minimize) in the top-left corner. Below the title bar is a row of five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons is a row of three input fields. The first field is labeled "Start Time" and contains the value "0". The second field is labeled "Stop Time" and contains the value "200". The third field is labeled "Period" and contains the value "10".

3. Enter the values for Start Time, Stop Time and Period.
4. Click OK.
5. Click on the *plot button* in the *Calculator* to plot the eyeDiagram.

## Flip Function

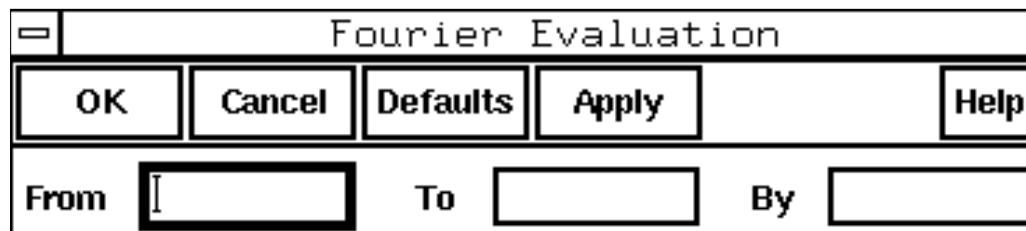
The *flip* function negates the X values. It has the effect of flipping the waveform horizontally in relation to the y axis.

## Fourier Evaluation (fourEval) Function

The *fourEval* function evaluates the Fourier series represented by the buffer expression. This function is an inverse Fourier transformation and thus the inverse of the *dft* function described in [Discrete Fourier Transform \(dft\) Function](#). It transforms the buffer expression from the frequency domain to the time domain.

1. Select *fourEval*.

The Fourier Evaluation form appears.



The screenshot shows a dialog box titled "Fourier Evaluation". It has a standard window control button (minimize) in the top-left corner. Below the title bar is a row of five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons is a row of three input fields. The first field is labeled "From" and contains the value "1". The second field is labeled "To" and is empty. The third field is labeled "By" and is empty.

2. Specify the time range over which you want to evaluate the series.

3. Enter the increment for evaluating the series.

## Frequency Function

The *frequency* function estimates the frequency of a periodic waveform. The system computes the reciprocal of the average time between two successive midpoint crossings of the rising waveform.

## Gain (gainBwProd/gainMargin) Functions

The *gainBwProd* function calculates the gain-bandwidth product. This function requires one argument, the frequency response of interest over a sufficiently large frequency range.

$$\text{gainBwProd}(\text{gain}) = A_o * f_2$$

The gain-bandwidth product is calculated as the product of the DC gain  $A_o$  and the critical frequency  $f_2$ . The critical frequency  $f_2$  is the smallest frequency for which the gain equals  $1/\sqrt{2}$  times the DC gain  $A_o$ .

The *gainMargin* function computes the dB value of the buffer expression when its phase crosses 180 degrees. This value represents the gain margin in unity gain configuration.

## Group Delay Function

The *groupDelay* function computes the group delay of the expression in the buffer.

Group delay is defined as the derivative of the phase with respect to frequency. Group delay is expressed in seconds. It is calculated using the *vp* function as shown below.

$$\text{Group Delay} = \frac{d\phi}{d\omega} = \frac{d}{df} \left[ \frac{\text{phase}(/netX)}{360} \right]$$

## Harmonic Function

This function returns the harmonic waveform of a waveform you specify.

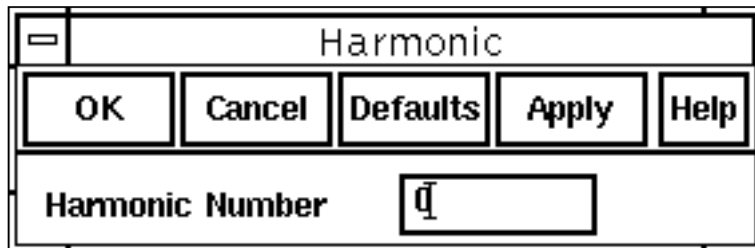
1. Enter the expression for the waveform in the Calculator buffer.

## Waveform Calculator User Guide

### RPN Mode

---

2. Choose *harmonic* in the *Special Functions* menu of the Calculator to open the Harmonic form.



Harmonic

OK Cancel Defaults Apply Help

Harmonic Number 1

3. In the Harmonic form, specify the harmonic you want by typing the value in the *Harmonic Number* field.
4. Click *OK* or *Apply* in the Harmonic form.
5. Click *plot* in the Calculator to plot the waveform.

To use this function, you must type the line below in the CIW

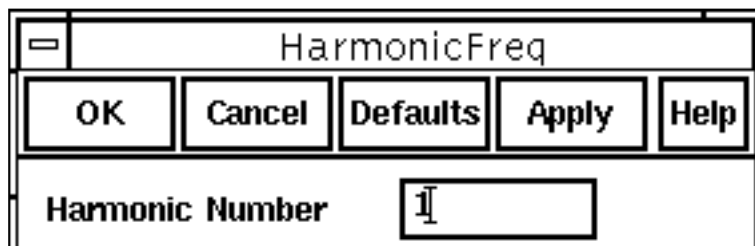
```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Harmonic Frequency Function

This function returns the harmonic waveform of a waveform you specify.

1. Enter the expression for the waveform in the Calculator buffer.
2. Choose *harmonicFreq* in the *Special Functions* menu of the Calculator to open the HarmonicFreq form.



HarmonicFreq

OK Cancel Defaults Apply Help

Harmonic Number 1

3. In the HarmonicFreq form, specify the harmonic you want by typing the value in the *Harmonic Number* field.
4. Click *OK* or *Apply*.

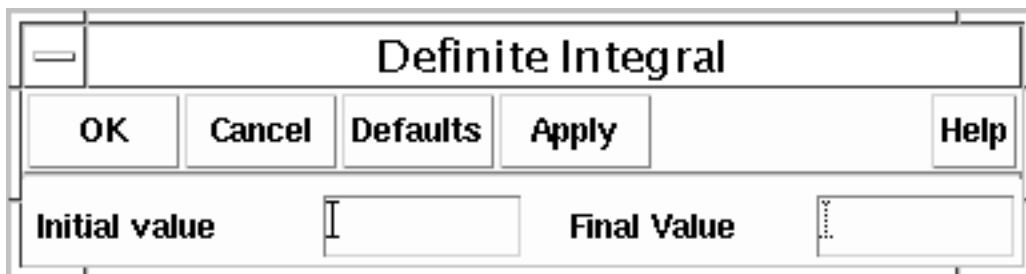
## iinteg Function

The *iinteg* integral function computes the indefinite integral of the buffer expression with respect to the X-axis variable. The result is a waveform that can be plotted.

## integ Function

The *integ* integral function computes the definite integral of the expression in the buffer. The result is the value of the area under the curve over a specified range on the X-axis of the expression.

1. Define a waveform in the buffer.
2. Choose *integ* from the *Special Functions* menu in Calculator. The Definite Integral form appears.



The image shows a dialog box titled "Definite Integral". It contains five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons are two input fields labeled "Initial value" and "Final Value". The "Initial value" field has a cursor inside it.

3. Enter the values for the limits of the definite integral in the *Initial Value* and *Final Value* fields.

**Note:** You should specify either both the limits or neither. In case you do specify the limits, they become the end points of the range on the X-axis for definite integration. If you do not specify the limits, then the range for definite integration is the entire range of the sweep on the X-axis.

4. Click *OK*.

## ipn Function

This function plots the *Nth* order intercept between two harmonics of a waveform that you define.

1. Enter the expression for the waveform in the Calculator buffer.

## Waveform Calculator User Guide

### RPN Mode

---

2. Choose *ipn* in the *Special Functions* menu of the Calculator. The IPN form opens.

IPN				
OK	Cancel	Defaults	Apply	Help
Spur Order	<input type="text" value="3"/>	Spur Harmonic	<input type="text" value="1"/>	
Extrapolation Point	<input type="text" value="0"/>	Reference Harmonic	<input type="text" value="1"/>	

3. Type values for the following four quantities:

Spur Order	Spur Order determines what order of interference is calculated for the spurious and reference waves. The default value is 3; this corresponds to the <i>IP3</i> function. If you use a value other than 3, that order of interference is calculated between those two waves.
Spur Harmonic	Harmonic number for spurious waveform.
Reference Harmonic	Harmonic number for reference waveform.
Extrapolation Point	The extrapolation point for the <i>IPN</i> function. This is the X axis value.

4. Click *OK* or *Apply*.

The expression is sent to the calculator buffer. To evaluate the expression, click the *Evaluate Buffer* button in the calculator.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## ipnVRI Function

This function performs an intermodulation Nth-order intercept point measurement.

Use this function to simplify the declaration of an ipn measurement. This function extracts the spurious and reference harmonics from the input waveform(s), and uses

`dBm(spectralPower((i or v/r),v))` to calculate the respective powers. The function

## Waveform Calculator User Guide

### RPN Mode

then passes these power curves or numbers and the remaining arguments to the *ipn* function to complete the measurement.

From each of the spurious and reference power waveforms (or points), the *ipn* function extrapolates a line of constant slope (dB/dB) according to the specified order and input power level. These lines represent constant small-signal power gain (ideal gain). The *ipn* function calculates the intersection of these two lines and returns the value of either the x coordinate (input referred) or y coordinate.

1. Enter the expression for the waveform in the Calculator buffer.
2. Choose *ipnVRI* in the *Special Functions* menu of the Calculator. The ipnVRI form opens.

The screenshot shows a dialog box titled "ipnVRI". At the top are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below these are several input fields and checkboxes. On the left side, there are three rows of labels and input boxes: "Spur Harmonic" with a box containing "1", "Spur Order" with a box containing "3", and "Load Resistance" with a box containing "50". On the right side, there are two rows of labels and input boxes: "Reference Harmonic" with a box containing "1", and "Extrapolation Point" with a box containing "1". Below these is a checkbox labeled "Input Referred IPN" which is checked. At the bottom, there is a label "Circuit Input Power is:" followed by a checkbox labeled "Variable Sweep" which is checked.

3. Type values for the following quantities:

*Spur Harmonic*                      Harmonic index for spurious waveform.

*Reference Harmonic*              Harmonic index for reference waveform.

*Spur Order*                          Spur Order determines what order of interference is calculated for the spurious and reference waves. The default value is 3; this corresponds to the *IP3* function. If you use a value other than 3, that order of interference is calculated between those two waves.

*Extrapolation Point*              The extrapolation point for the *ipn* function. This is the X axis value. The default is the minimum x value of the input voltage waveform.

## Waveform Calculator User Guide

### RPN Mode

#### *Load Resistance*

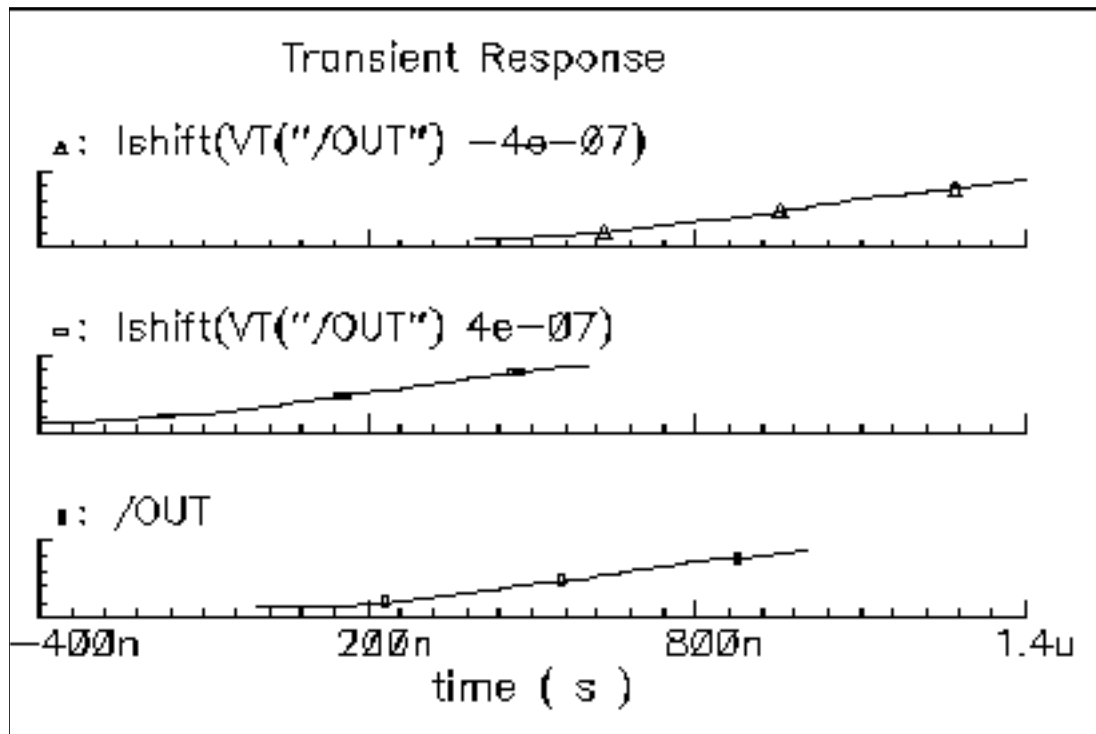
The resistance into the output port. The default value is 50.

4. To get the X-coordinate of the intercept, specify *Input Referred IPN*. To get the Y-coordinate of the intercept, specify *Output Referred IPN*.
5. Indicate whether the *Circuit Input Power* is a *Variable Sweep* or a *Single Point*.
6. Click *OK* or *Apply*.

**Note:** For an extended design example, using *ipn* and *ipnVRI* functions, see [Appendix A, "Using the Calculator Special Functions with SpectreRF Simulation Results"](#).

## Lshift Function

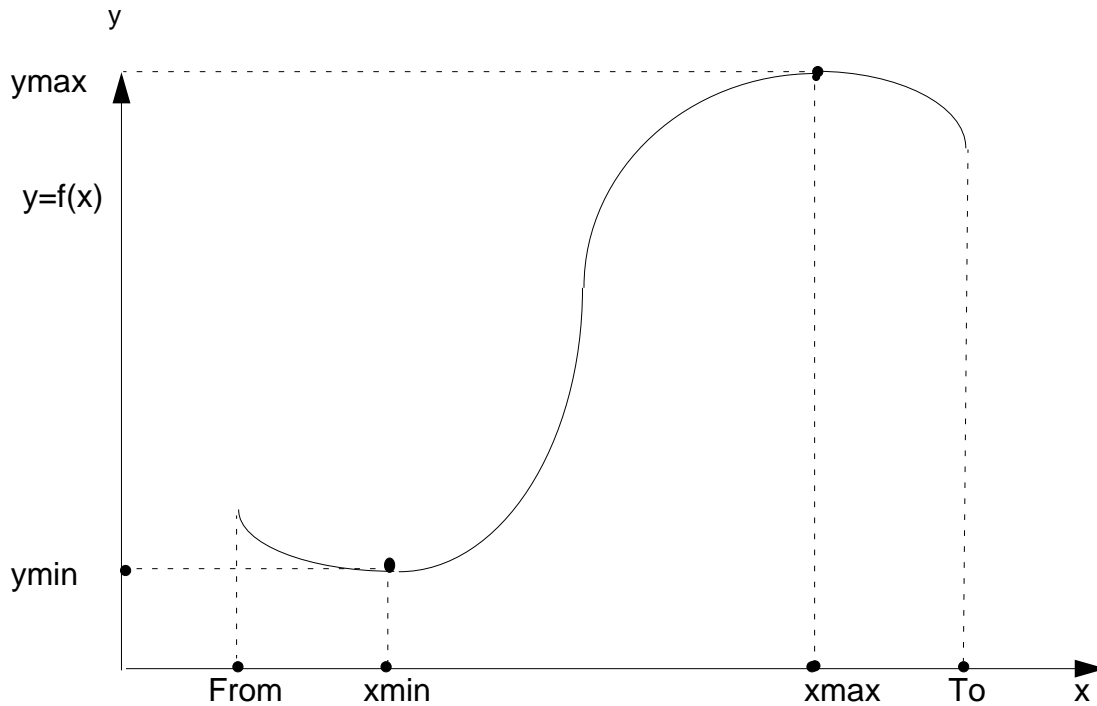
The *Lshift* function shifts the data in the Waveform window to the left by a specified amount. A negative value shifts the data to the right.



## Minimum and Maximum Functions

You can calculate minimum and maximum values of waveforms with the *xmax*, *xmin*, *ymin*, and *ymin* functions.

This figure shows the relationship of these functions.



### **xmax and ymax**

The *xmax* function computes the value of the independent variable  $x$  at which the expression attains its maximum value, that is, the value of  $x$  that maximizes  $y=f(x)$ .

The maximum might occur at more than one point on the  $x$  axis, so you must choose (in the *Nth Maximizer* field) which maximum value you want to see. The calculator returns the value of the *Nth* Maximizer counting from the left, that is, toward increasing  $X$ -axis values. If you

## Waveform Calculator User Guide

### RPN Mode

---

enter a negative integer, the direction of search is reversed toward decreasing X-axis values (counting from the right).

=	X Value at Maximum			
OK	Cancel	Defaults	Apply	Help
Nth Maximizer		<input type="text" value="1"/>		

The *y*max function computes the maximum y of the expression  $y=f(x)$ .

#### xmin and ymin

The *x*min function computes the value of the independent variable x at which the expression has its minimum value, that is, the value of x that minimizes  $y=f(x)$ .

The minimum might occur at more than one point on the x axis, so you must choose (in the *Nth Minimizer* field) which minimum value you want to see. The calculator returns the value of the Nth Minimizer, counting from the left, that is, toward increasing X-axis values. If you enter a negative integer, the direction of search is reversed toward decreasing X-axis values (counting from the right).

=	X Value at Minimum			
OK	Cancel	Defaults	Apply	Help
Nth Minimizer		<input type="text" value="1"/>		

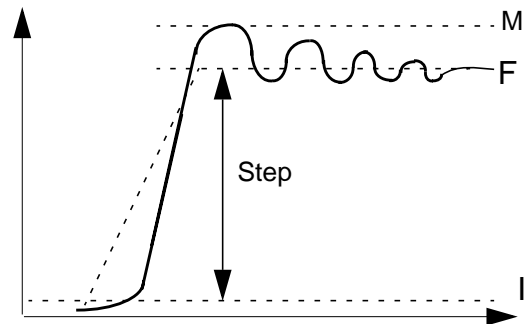
The *y*min function computes the minimum y of the expression  $y=f(x)$ .

## Overshoot Function

The *overshoot* function computes the percentage by which the buffer expression overshoots a step going from the Initial Value to the Final Value you enter.

Overshoot			
OK	Cancel	Defaults	Apply
Initial Value	◆ y at x ◇ y	<input style="width: 100%;" type="text"/>	
Final Value	◆ y at x ◇ y	<input style="width: 100%;" type="text"/>	

$$Overshoot = \frac{M - F}{F - I}$$

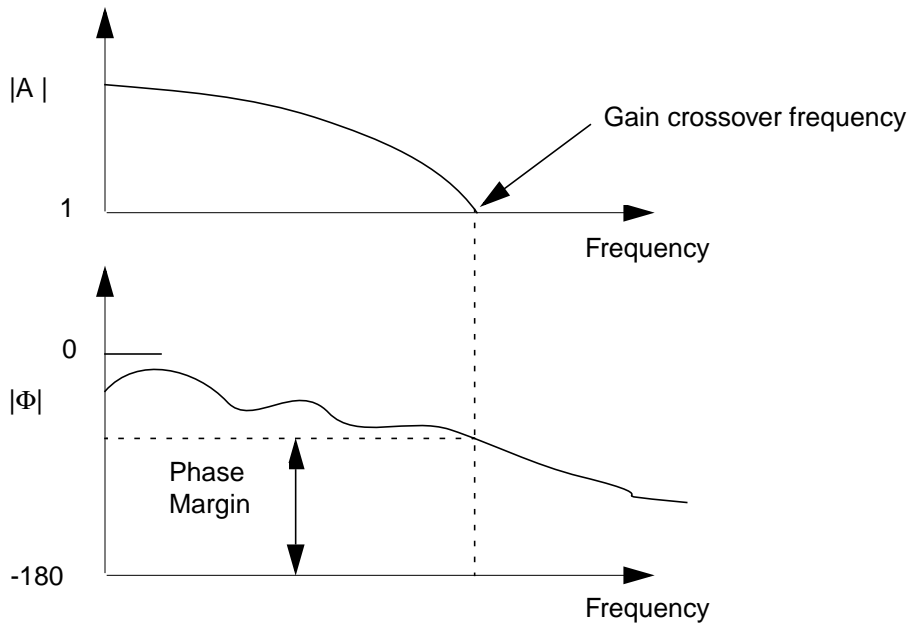


There are two ways to set the Initial and Final Values:

- Use the *y* option and enter the *y* values directly (for example, 3V) or a horizontal [marker](#) name (for example, M2).
- Use the *y at x* option to enter an *x* value or vertical marker name (for example, M1). The system calculates the corresponding *y* value for the current waveform.

## Phase Margin Function

The *phaseMargin* function computes the difference between the phase (in degrees) at 180 and the frequency at which the buffer expression magnitude equals 1. This function is similar to the [phaseDegUnwrapped](#) SKILL command.



## Phase Noise Function

This function plots the phase noise waveform for noise analysis results. You need to follow the following steps to use this function.

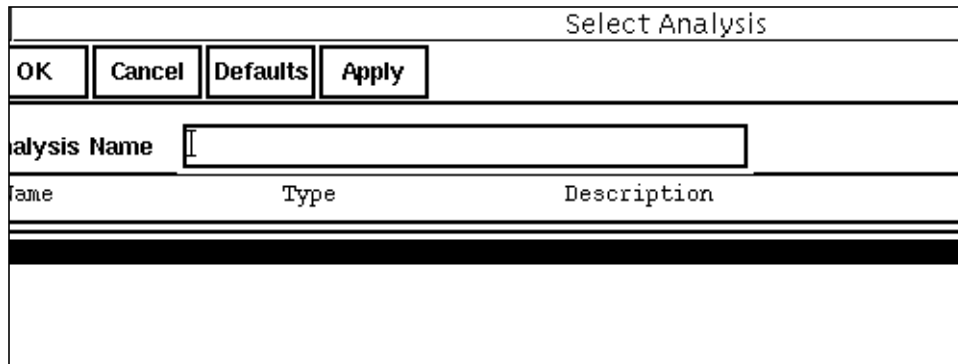
1. Set up your PSS analysis.
2. Check the *oscillator button* in the *choosing analysis* form.
3. Set up a Pnoise analysis. Note the value you are entering for the relative harmonic in the *choosing analysis* form.
4. Run your simulation.

## Waveform Calculator User Guide

### RPN Mode

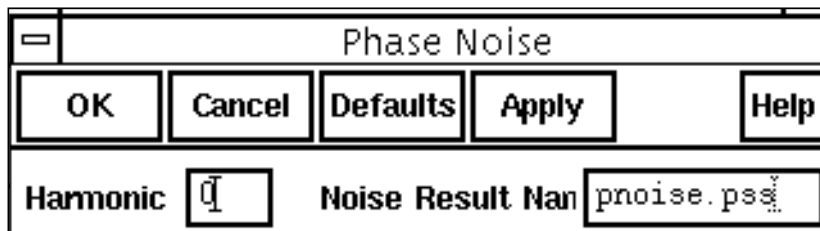
---

5. Invoke the Calculator . Choose *phaseNoise* in the *Special Functions* menu of the Calculator. The *Select Analysis* form appears.



The *Select Analysis* dialog box has a title bar with the text "Select Analysis". Below the title bar are four buttons: "OK", "Cancel", "Defaults", and "Apply". Below the buttons is a text field labeled "Analysis Name" with a cursor inside. Below the text field is a table with three columns: "Name", "Type", and "Description". The table is currently empty.

6. Choose the pnoise-pnoise analysis name in the *Analysis Name* field of the *Select Analysis* form, or select the analysis name from the list box in the form. Click on the *OK* button in the form. The *Phase Noise* form appears.



The *Phase Noise* dialog box has a title bar with the text "Phase Noise". Below the title bar are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below the buttons are two text fields: "Harmonic" with a cursor inside, and "Noise Result Name" with the text "pnoise.pss" inside.

7. In the Phase Noise form, enter *harmonic number*=1 or the value you had entered for relative harmonic in the *pnoise choosing analysis* form.
8. Enter the *Noise Result Name* as pnoise-pnoise and click on *OK* in the form.
9. In your calculator buffer, you should see something like `phaseNoise(1, "pss-fd.pss", "result "pnoise-pnoise")`.
10. Click on the *plot* button in the Calculator to plot the waveform.
11. The phase noise plot will appear.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Power Spectral Density (psd) Function

The power spectral density (*psd*) function describes how the power (or variance) of a time series (signal) is distributed with frequency. Mathematically, it is defined as the Fourier Transform of the auto correlation sequence of the time series (signal). The waveform is first interpolated, to generate evenly spaced data points in time. The spacing of the data points is the inverse of the *dft* sampling frequency. The *psd* is computed by first breaking up the time interval into overlapping segments. Each segment is multiplied, time point by time point, by the specified windowing function. The *dft* is performed on each windowed segment of the baseband waveform. At each frequency, the *dfts* from all segments are averaged together and the squared modulus of these averages, gives *psd*.

After you choose *psd* in the *Special Functions* menu of the Calculator, the Power Spectral Density form opens.

Power Spectral Density				
OK		Cancel	Defaults	Apply
Help				
From	<input type="text"/>	To	<input type="text"/>	Number of Samples <input type="text" value="512"/>
Window Type	<input type="text" value="Hanning"/>	Smooth. Fac.	<input type="text" value="1"/>	Window Size <input type="text" value="256"/>
Coherent Gain	<input type="text" value="(none)"/>	<input type="text" value="1"/>	Detrending Mode	<input type="text" value="None"/>

1. In the *From* field, type the starting time for the spectral analysis interval.
2. In the *To* field, type the ending time for the spectral analysis interval. You can use this parameter and the *From* parameter to exclude part of the interval. For example, you might set these values to discard initial transient data.
3. In the *Number of Samples* field, type the number of time domain points to use. The maximum frequency in the Fourier analysis is proportional to the *Number of Samples* parameter and inversely proportional to the difference between the starting time and the ending time.
4. Choose the *Window Type* that you want to use. If you select the *Kaiser* window type, then type in a value for the Kaiser smoothing factor. The smoothing factor must be in the range  $0 \leq \text{factor} \leq 15$ , where 0 is the same as using a rectangular window.
5. In the *Window Size* field, type in the number of frequency domain points to use in the Fourier analysis.

## Waveform Calculator User Guide

### RPN Mode

A larger window size results in an expectation operation over fewer samples, which leads to larger variations in the power spectral density. A small window size can smear out sharp steps in the power spectral density that might really be present.

6. Choose a *Coherent Gain* factor. If you choose *magnitude*, *dB20*, or *dB10*, then enter a scaling factor. A non-zero factor scales the power spectral density by  $1/(\text{factor})$ . Valid values for the factor are  $0 < \text{factor} < 1$ . You can also use a value of 1 if you do not want the *Coherent Gain* factor to be used.

7. Choose which *Detrending Mode* to use.

The *psd* function works by applying a moving windowed FFT to time-series data. If there is a deterministic trend to the underlying data, you might want to remove the trend before performing the spectral analysis. For example, consider analyzing phase noise in a VCO model. Without the noise the phase increases more or less linearly with time, so it is appropriate to set the detrending mode to *linear*. To subtract an average value, set the detrending mode to *mean*. Where the spectrum of raw data is desired, set the detrending mode to *none*.

8. Click *OK*.

## Power Spectral Density Baseband (psdbb) Function

The power spectral density baseband (*psdbb*) function returns an estimate for the power spectral density of a  $\text{waveform1} + j * \text{waveform2}$ .

After you choose *psdbb* in the *Special Functions* menu of the Calculator, the Power Spectral Density Baseband form opens.

The screenshot shows a dialog box titled "Power Spectral Density Baseband". It contains several controls: "OK", "Cancel", "Defaults", "Apply", and "Help" buttons at the top. Below these are input fields for "From" and "To" (empty), "Number of Samples" (512), "Window Type" (Hanning with a dropdown arrow), "Smooth. Fac." (1), "Window Size" (256), "Coherent Gain" ((none) with a dropdown arrow), and "Detrending Mode" (None with a dropdown arrow).

1. In the *From* field, type the starting time for the spectral analysis interval.
2. In the *To* field, type the ending time for the spectral analysis interval. You can use this parameter and the *From* parameter to exclude part of the interval. For example, you might set these values to discard initial transient data.

## Waveform Calculator User Guide

### RPN Mode

---

3. In the *Number of Samples* field, type the number of time domain points to use. The maximum frequency in the Fourier analysis is proportional to the *Number of Samples* parameter and inversely proportional to the difference between the starting time and the ending time.
4. Choose the *Window Type* that you want to use. If you select the *Kaiser* window type, then type in a value for the Kaiser smoothing factor. The smoothing factor must be in the range  $0 \leq \text{factor} \leq 15$ , where 0 is the same as using a rectangular window.
5. In the *Window Size* field, type in the number of frequency domain points to use in the Fourier analysis.

A larger window size results in an expectation operation over fewer samples, which leads to larger variations in the power spectral density. A small window size can smear out sharp steps in the power spectral density that might really be present.

6. Choose a *Coherent Gain* factor. If you choose *magnitude*, *dB20*, or *dB10*, then enter a scaling factor. A non-zero factor scales the power spectral density by  $1/(\text{factor})$ . Valid values for the factor are  $0 < \text{factor} < 1$ . You can also use a value of 1 if you do not want the *Coherent Gain* factor to be used.
7. Choose which *Detrending Mode* to use.

The *psddb* function works by applying a moving windowed FFT to time-series data. If there is a deterministic trend to the underlying data, you might want to remove the trend before performing the spectral analysis. For example, consider analyzing phase noise in a VCO model. Without the noise the phase increases more or less linearly with time, so it is appropriate to set the detrending mode to *linear*. To subtract an average value, set the detrending mode to *mean*. Where the spectrum of raw data is desired, set the detrending mode to *none*.

8. Click *OK*.

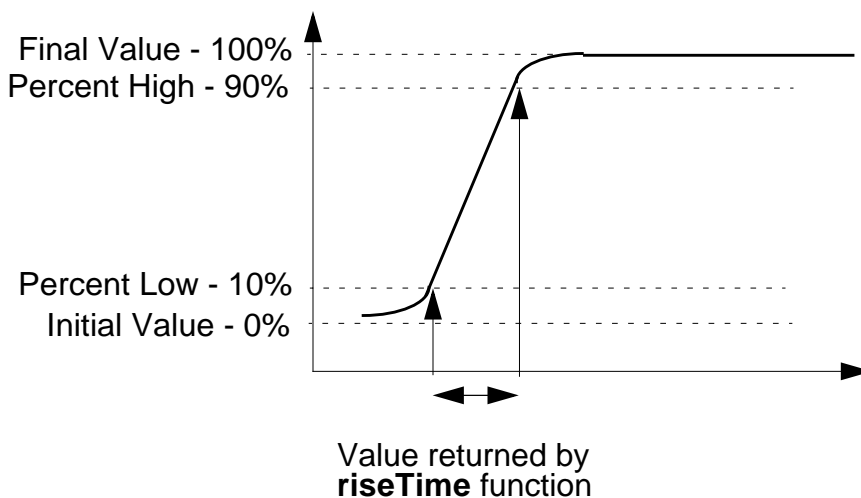
## Rise Time Function

The *riseTime* function computes the rise time of the buffer expression, that is, the time required to change from the value in the *Percent Low* field to the value in the *Percent High* field of the difference between Initial Value and Final Value.

Rise Time			
OK	Cancel	Defaults	Apply
Initial Value    ◇ y at x    ◆ y <input style="width: 100px;" type="text"/>			
Final Value    ◇ y at x    ◆ y <input style="width: 100px;" type="text"/>			
Percent Low <input style="width: 50px;" type="text" value="10"/>		Percent High <input style="width: 50px;" type="text" value="90"/>	

To set the initial and final values in the Rise Time form you can:

- Use the y option for "Initial Value" and "Final Value" and enter the y values directly or a horizontal [marker](#) name.
- Use the y at x option for "Initial Value" and "Final Value" to enter an x value or vertical [marker](#) name.



The above graph represents the "Initial value" of the signal as 0% and "Final value" as 100%. The "Percent Low" and "Percent High" values are taken as 10% and 90% .

For waveforms with multiple rise and fall edges, it is suggested that users should isolate edges of interest by using the clip() function or enter values for y at x rather than entering y values directly in the Rise Time function input form.

## Root-Mean-Square (rms) Function

The *rms* function computes the root-mean-square value of the expression  $f(x)$ , over the specified range of  $x$ . This is the square root of the integral of the expression squared over the specified range, divided by the range.

For example, if  $y = f(x)$ ,

$$rms(y) = \sqrt{\frac{\int_{from}^{to} f(x)^2 dx}{to - from}}$$

To compute the rms value of the expression over a smaller range, use the [clip function](#) inside the *rms* function.

## Root-Mean-Square (rms) Noise Function

The *rmsNoise* function computes the integrated root-mean-square of the total output noise over the bandwidth specified in hertz in the *From* and *To* fields.

RMS Noise				
OK	Cancel	Defaults	Apply	Help
From		To		

**Note:** To plot the squared noise voltage versus frequency instead, use the [Results – Plot Noise command](#) in the Simulation window.

## Root Function

The *root* function computes the value of  $x$  at which  $f(x)$  equals the specified threshold.

## Waveform Calculator User Guide

### RPN Mode

---

1. Select *root*.

The Root Value form appears.

Root Value				
OK	Cancel	Defaults	Apply	Help
Threshold Value	2.5	Nth Root	1	

2. In *Threshold Value*, enter the waveform value at which to compute the root value.
3. Enter the *root* you want to see.

## Sample Function

The *sample* function samples a waveform at the interval you specify. You can use this function to speed up plotting of waveforms that have many data points.

1. Define a waveform in the calculator buffer.
2. Select *sample*.

The Sample form appears.

Sample						
OK	Cancel	Defaults	Apply	Help		
From		To	1	linear 1L	By	

3. Specify the range and increment.

If you sample a waveform beyond its range, you get the final value of the waveform. You can use this function to demodulate a signal. Consider an AM modulated sine wave. Assume the carrier frequency is 1 GHz, and the modulation frequency is 1 MHz. If the waveform is sampled every 1 ns, the resulting signal will be cleanly demodulated (the 1 GHz carrier is completely eliminated by the sampling).

## Settling Time Function

The *SettlingTime* is the time by which the signal settles within the specified "Percent of step" of the difference between the "Final Value" and "Initial Value" from the "Final Value".

1. Select *settlingTime*. The Settling Time form appears.

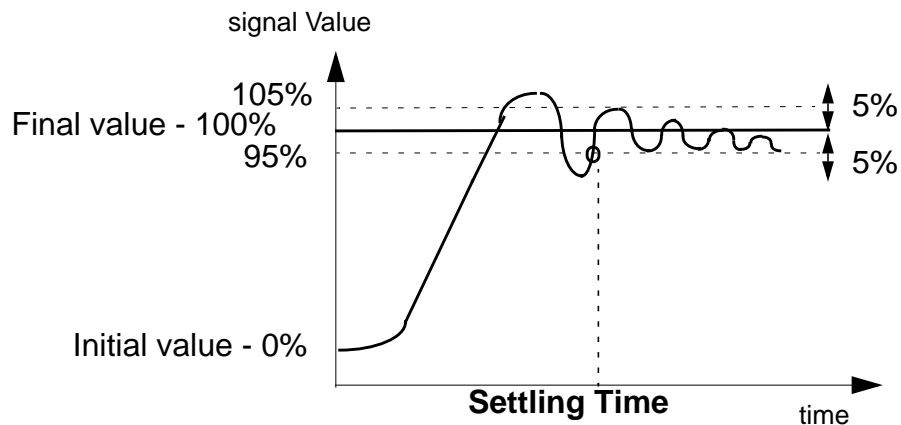
Settling Time				
OK	Cancel	Defaults	Apply	Help
Initial Value	◆ y at x ◇ y	<input style="width: 100%;" type="text"/>		
Final Value	◆ y at x ◇ y	<input style="width: 100%;" type="text"/>		
Percent of Step	<input style="width: 100%;" type="text" value="5"/>			

2. To set the initial and final values in the Settling Time form you can:

- Use the *y* option for "Initial Value" and "Final Value" and enter the *y* values directly or a horizontal [marker](#) name.
- Use the *y at x* option for "Initial Value" and "Final Value" to enter an *x* value or vertical [marker](#) name.

3. Enter the value for *Percent of Step*.

4. Click OK.



## Waveform Calculator User Guide

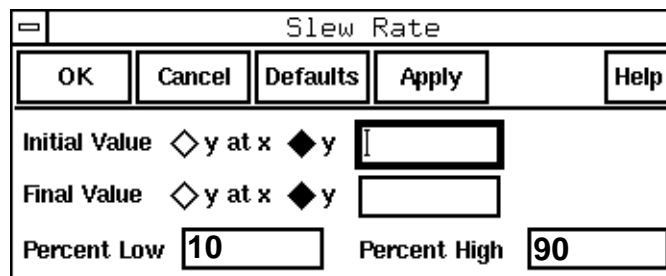
### RPN Mode

---

The above graph represents the “Initial value” of the signal as 0% and “Final value” as 100%. The “Percent of Step” is taken as 5%.

## Slew Rate Function

The *slewRate* function computes the average rate at which the buffer expression changes from percent low to percent high of the difference between initial value and final value.



Slew Rate				
OK	Cancel	Defaults	Apply	Help
Initial Value	<input type="radio"/> y at x	<input type="radio"/> y	<input type="text"/>	
Final Value	<input type="radio"/> y at x	<input type="radio"/> y	<input type="text"/>	
Percent Low	<input type="text" value="10"/>	Percent High	<input type="text" value="90"/>	

To set the initial and final values, do one of the following:

- Use the *y* option for "Initial Value" and "Final Value" and enter the *y* values directly or a horizontal [marker](#) name.
- Use the *y at x* option for "Initial Value" and "Final Value" to enter an *x* value or vertical [marker](#) name.

For waveforms, with multiple rise and fall edges, it is recommended that users isolate edges of interest by using the *clip()* function or enter values for *y at x* rather than entering *y* values directly in the Slew Rate function input form.

## Spectral Power Function

This function plots the spectral power for a current waveform and a voltage waveform that you define. To use this function:

1. Define the voltage waveform.
2. Define the current waveform.

When you specify the second waveform, the first waveform is passed to the stack.

3. Choose *spectralPower* in the *Special Functions* menu of the Calculator.
4. Click *plot* in the Calculator to plot the waveform.

## Waveform Calculator User Guide

### RPN Mode

---

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

### Standard Deviation (`stddev`) Function

The *stddev* function computes the standard deviation of a waveform (or a family of waveforms) over its entire range. Standard deviation (`stddev`) is defined as the square-root of the variance where variance is the integral of the square of the difference of the expression  $f(x)$  from average ( $f(x)$ ), divided by the range of  $x$ .

For example, if  $y=f(x)$

$$stddev(y) = \sqrt{\frac{\int_{from}^{to} (y - average(y))^2}{to - from}}$$

If you want a different range, use the [clip function](#) to clip the waveform to the range you want.

### Table Function

The *table* function defines a piecewise linear function from a column of  $x$  and  $y$  values in a file. This function was previously named `implicitX`.

1. Enter the name of the data file and any name for the function.
2. (Optional) Enter the column numbers containing the  $X$  and  $Y$  axis data, if they are not in columns 1 and 2 respectively.
3. (Optional) Enter the number of lines to skip in each column from the top of the file before reading the data.

## Waveform Calculator User Guide

### RPN Mode

**Note:** Do not count comment lines beginning with a semicolon and blank lines in the number of lines to skip.

Table						
OK	Cancel	Defaults	Apply			Help
File Name	<input type="text"/>					
Function Name	<input type="text"/>					
X Column Number	<input type="text" value="1"/>	Y Column Number	<input type="text" value="2"/>			
X Skip Lines	<input type="text" value="0"/>	Y Skip Lines	<input type="text" value="0"/>			

The X data must be real numbers increasing monotonically. The Y data can be real numbers, or complex numbers following this syntax:

`(real_part imag_part)`

or

`complex(real_part imag_part)`

## Tangent Function

This function plots a line that passes through x and y coordinates and the slope that you specify. To use this function:

1. Define a waveform in the buffer.
2. Choose *tangent* from the *Special Functions* menu in the Calculator. The Tangent Line form appears.

Tangent Line					
OK	Cancel	Defaults	Apply	Help	
X point	<input type="text" value="0"/>	Y Point	<input type="text" value="0"/>	Slope	<input type="text" value="1.0"/>

3. Type values for the *X point*, *Y Point*, and *Slope* fields.

## Waveform Calculator User Guide

### RPN Mode

4. Click *OK*.
5. Click *plot* in the Calculator to plot the tangent line.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Total Harmonic Distortion (thd) Function

The *thd* function computes the percentage of total harmonic content of a signal with respect to the fundamental frequency. The computation uses the *dft* function (for information, see [Discrete Fourier Transform \(dft\) Function](#)). Assume that the *dft* function returns complex coefficients  $A_0, A_1, \dots, A_f, \dots$ . Please note that fundamental frequency  $f$  is the frequency contributing to the largest power in the signal.  $A_0$  is the complex coefficient for the DC component and  $A_i$  is the complex coefficient for the  $i$ th harmonic where  $i \neq 0, f$ . Then, total harmonic distortion is computed as:

$$\frac{\sqrt{\sum_{i=1, i \neq 0, f} |A_i|^2}}{|A_f|} \times 100 \%$$

To compute the *thd*, you need to perform the following steps:

1. Choose *thd* in the *Special Functions* menu of the calculator.

The Total Harmonic Distortion form appears.

Total Harmonic Distortion				
OK		Cancel	Defaults	Apply
From		To	Number of Samples	
<input type="text"/>		<input type="text"/>	<input type="text" value="64"/>	
Fundamental (Hz)		<input type="text"/>	Enter 0 to choose the largest signal	

2. Specify the range and the number of samples.
3. Click *OK*.
4. Click *Print* to see the result.

## Waveform Calculator User Guide

### RPN Mode

The accuracy of the total harmonic distortion measurement depends on simulator options and the analysis parameters. For an accurate measurement set the following simulation options:

Option	Suggested Value
RELTOL	1e-5
ABSTOL	1e-13
VNTOL	3e-8
TRTOL	1
METHOD	gear
MAXORD	3

Set the simulation timestep to be 1/100th of a cycle, and simulate for ten cycles. End the simulation slightly beyond the tenth cycle. When you use the calculator, measure during the tenth cycle by specifying the beginning of the cycle as the From time and the end as the To time.

### Value Function

The *value* function computes the value of the waveform at the point you specify.

- Enter the point at which to compute the value.

=	Value			
OK	Cancel	Defaults	Apply	Help
Interpolate At		<input type="text"/>		

### X Value (xval) Function

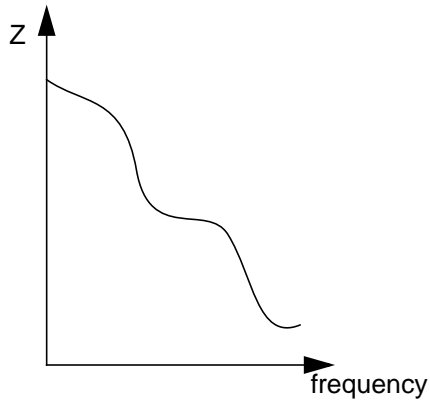
The *xval* function takes a single expression as an argument. It returns a waveform with the Y values equal to the X values:  $Y=X$ . This facilitates computations where the dependent

## Waveform Calculator User Guide

### RPN Mode

---

variable (such as time or frequency) is needed in an expression. For example, you can use `xval` to compute the capacitance waveform for this curve:



$$\text{Capacitance Waveform } C(f) = \frac{1}{2\pi Z \cdot xval(Z)}$$

Here are some syntax examples:

```
xval(VF("/IN"))
```

```
xval(VT("/net3"))
```

1. Set up the expression in the calculator buffer.
2. Select `xval`.

## **Waveform Calculator User Guide**

### **RPN Mode**

---

---

## Algebraic Mode

This section describes how you use the calculator in algebraic mode.

### Operators and Functions

The calculator has both algebraic and Reverse Polish Notation (RPN) modes.

**Note:** Each of the calculator functions has a corresponding SKILL command. For more information about SKILL calculator functions, refer to the [OCEAN Reference](#).

In algebraic mode you build expressions from left to right. Whenever you click an operator or function key, the operator or function is added to the right side of the expression in the buffer. The system adds left parentheses automatically when necessary. You must enter the corresponding closing parentheses.

For example, to enter the function  $(1+x)/x$  in RPN mode, you use this key sequence:

1 enter clear x + lastx /

To enter  $(1+x)/x$  in algebraic mode, you use this key sequence:

1 + x ( ) / x

Help is also available for

- The [buffer](#) and [stack](#)
- RPN mode

### Single-Expression Functions

The following functions operate on only a single expression in the buffer.

Key	Function	Key	Function
<i>mag</i>	magnitude	<i>exp</i>	$e^x$

## Waveform Calculator User Guide

### Algebraic Mode

Key	Function	Key	Function
<i>phase</i>	phase (wrapped, in degrees)	$10^{**}x$	$10^x$
<i>real</i>	real component	$y^{**}x$	$y^x$
<i>imag</i>	imaginary component	$x^{**}2$	$x^2$
<i>ln</i>	base-e (natural) logarithm	<i>abs</i>	x  (absolute value)
<i>log10</i>	base-10 logarithm	<i>int</i>	integer value
<i>dB10</i>	dB magnitude for a power expression	$1/x$	inverse
<i>dB20</i>	dB magnitude for a voltage or current	<i>sqrt</i>	$\sqrt{x}$

**Note:** Selecting these functions while the buffer contains multiple expressions is an error in RPN mode because there is a space between expressions. For example,  $\ln(\text{expr1} \text{ expr2})$  is invalid because the logarithm function takes only one argument, not two.

### Example: Plotting the Magnitude of a Signal

To plot the dB magnitude of a signal after an AC analysis in algebraic mode

1. Click *dB20* on the calculator.
2. Click *vf* on the calculator.
3. On the schematic, click the net you want to plot.
4. With the cursor in the Schematic window, press the *Esc* key.  
This cancels the *vf* function. Otherwise, the command stays active.
5. Press the right parenthesis key twice.  
The calculator buffer now contains the expression you want to plot.
6. Click *plot* to show the curve.

### Two-Expression Functions and Operators

To use the two-expression functions and arithmetic operators

## Waveform Calculator User Guide

### Algebraic Mode

---

1. Enter the first operand.
2. Choose the function or operator.
3. Enter the second operand.

---

Key	Function in algebraic mode
$y^{**}x$	$y^x$
+	Adds the buffer expression to the first stack register.
-	Subtracts the buffer expression from the first stack register.
*	Multiplies the buffer expression by the first stack register.
/	Divides the first stack register by the buffer expression.

---

### Example: Instantaneous Power Dissipation

This example computes the instantaneous power dissipated by a resistor.

1. Click *vt*.
2. On the schematic, click the net connected to the appropriate pin of the resistor.
3. With the cursor in the Schematic window, press the *Esc* key.

This cancels the *vt* function. Otherwise, the command stays active.

4. Click *\** on the calculator.
5. Click *it*.
6. Click the appropriate pin of the resistor and then press *Esc*.

To select currents, click the square pin symbol. Do not click the wire stub.

7. Click *plot*.

## Trigonometric Functions

The trigonometric functions work like the other single-expression functions. They are appended to the right of the buffer, rather than operating on the buffer.

1. Click a trigonometric function key.

The system appends the function and a left parenthesis to the right side of the buffer.

## Waveform Calculator User Guide

### Algebraic Mode

---

2. Enter the operand expression you want to operate on.
3. Enter a right parenthesis.

sin	asin
cos	acos
tan	atan
sinh	asinh
cosh	acosh
tanh	atanh

## Special Functions

The special functions help you analyze waveform data generated with calculator expressions. Some functions in the *Special Functions* menu, pop up a form where you enter the data required for the calculation. Other special functions are appended to the buffer.

<i>xmax</i>	<i>fourEval</i>	<i>psdbb</i>
<i>xmin</i>	<i>frequency</i>	<i>riseTime</i>
<i>ymax</i>	<i>gainBwProd</i>	<i>rms</i>
<i>ymin</i>	<i>gainMargin</i>	<i>rmsNoise</i>
<i>average</i>	<i>groupDelay</i>	<i>root</i>
<i>bandwidth</i>	<i>harmonic</i>	<i>sample</i>
<i>clip</i>	<i>harmonicFreq</i>	<i>settlingTime</i>
<i>compression</i>	<i>iinteg</i>	<i>slewRate</i>
<i>compressionVRI</i>	<i>integ</i>	<i>spectralPower</i>
<i>convolve</i>	<i>ipn</i>	<i>stddev</i>
<i>cross</i>	<i>ipnVRI</i>	<i>table</i>
<i>dBm</i>	<i>lshift</i>	<i>tangent</i>
<i>delay</i>	<i>overshoot</i>	<i>thd</i>

## Waveform Calculator User Guide

### Algebraic Mode

---

<i>deriv</i>	<i>phaseMargin</i>	<i>value</i>
<i>dft</i>	<i>phaseNoise</i>	<i>xval</i>
<i>flip</i>	<i>psd</i>	

---

### Example: Average Value of a Current

This example shows how to compute the average of a current during the simulation period.

1. Choose *Special Functions – Average*.
2. Click *it* on the calculator.
3. On the schematic, click the terminal whose current you want to average.  
To select currents, click the square pin symbol. Do not click the wire stub.
4. With the cursor in the Schematic window, press the *Esc* key.  
This cancels the *it* function. Otherwise, the command stays active.
5. Enter a right parenthesis.
6. Click *print* on the calculator.  
The system displays the average value in a text window.

### Example: Delay Function

This example shows how to append a delay special function to the calculator buffer, assuming that an expression is already in the buffer.

1. Choose *delay* from the *Special Functions* menu.  
The buffer is saved to the memory *asfmem* and then cleared.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Threshold Delay form appears.

2. To build the first expression for the delay function, do the following:

- a. Click *vt*.
- b. Click a node in the schematic.

The expression `vt("/net1")` now appears in the buffer.

3. Click *Wf1: Get Buffer* to import the buffer expression into the Threshold Delay form.

4. Specify the threshold values, edge numbers, and edge type for the first expression.

5. To build the second waveform in the buffer, do the following:

- a. Click *clear* to clear the first expression from the buffer.
- b. The *vt* function is still active. You can now click a node in the schematic.

The expression `vt("/net2")` now appears in the buffer.

6. Click *Wf2: Get Buffer* to import the second expression.

7. Specify the threshold values, edge numbers, and edge type for the second expression.

8. Click *OK* in the Threshold Delay form.

The original expression in the buffer is recalled, and the delay function is appended to the right of this expression.

```
orig_expression delay(VT("/net1"),2.5,1,"either",  
VT("/net2"),2.5,1,"either")
```

## Algebraic Mode Special Functions

### Average Function

The *average* function computes the average of a waveform over its entire range. Average is defined as the integral of the expression  $f(x)$  over the range of  $x$ , divided by the range of  $x$ . For example, if  $y=f(x)$ ,  $\text{average}(y) =$

$$\frac{\int_{\text{from}}^{\text{to}} f(x)dx}{\text{to} - \text{from}}$$

where *to* and *from* are the range of  $x$ .

If you want a different range, use the [clip function](#) to clip the waveform to the range you want.

### Bandwidth Function

The *bandwidth* function calculates the bandwidth of the waveform in the calculator buffer. Please note that the input waveform must represent a true voltage, NOT modified by a dB.

1. Set up the expression whose bandwidth you want to calculate in the buffer.
2. Select *bandwidth*.

The Bandwidth form appears.

Bandwidth				
OK	Cancel	Defaults	Apply	Help
Get Buffer	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>			
Db	<div style="border: 1px solid black; width: 100px; height: 20px; display: flex; align-items: center; justify-content: center;">3</div>	Type	<div style="border: 1px solid black; display: inline-block; padding: 2px 5px;">low</div> <input type="checkbox"/>	

3. Click *Get Buffer*.
4. In the *Db* field, enter how far below the peak value you want to see data.
5. Choose a bandwidth response from the *Type* field.

## Waveform Calculator User Guide

### Algebraic Mode

---

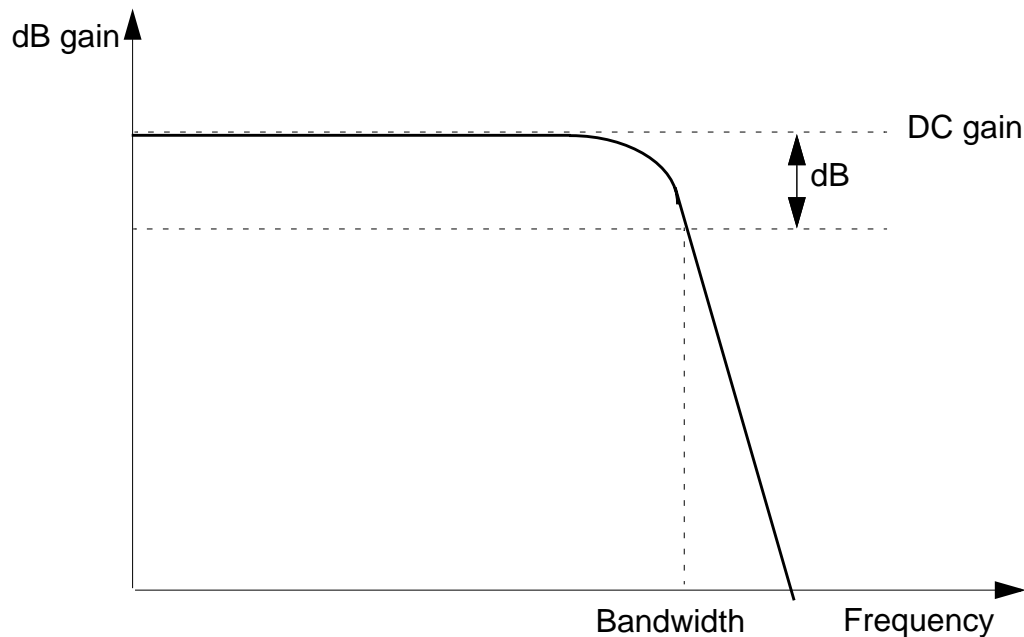
- ☐ [low](#) computes the bandwidth of a low-pass response.
- ☐ [high](#) computes the bandwidth of a high-pass response.
- ☐ [band](#) computes the bandwidth of a band-pass response.

6. Click *OK*.

### Computing Low-Pass Bandwidth

The calculator computes the low-pass bandwidth by determining the smallest frequency at which the magnitude of the input waveform drops  $n$  decibels below the DC gain. (DC gain is obtained by zero-order extrapolation from the lowest or highest computed frequency, if necessary.) The *dB* field specifies  $n$ . An error occurs if the magnitude of the input waveform does not drop  $n$  decibels below the DC gain.

#### Low-Pass Bandwidth Function



### Computing High-Pass Bandwidth

The calculator computes the high-pass bandwidth by determining the largest frequency at which the magnitude of the input waveform drops  $n$  decibels below the gain at the highest

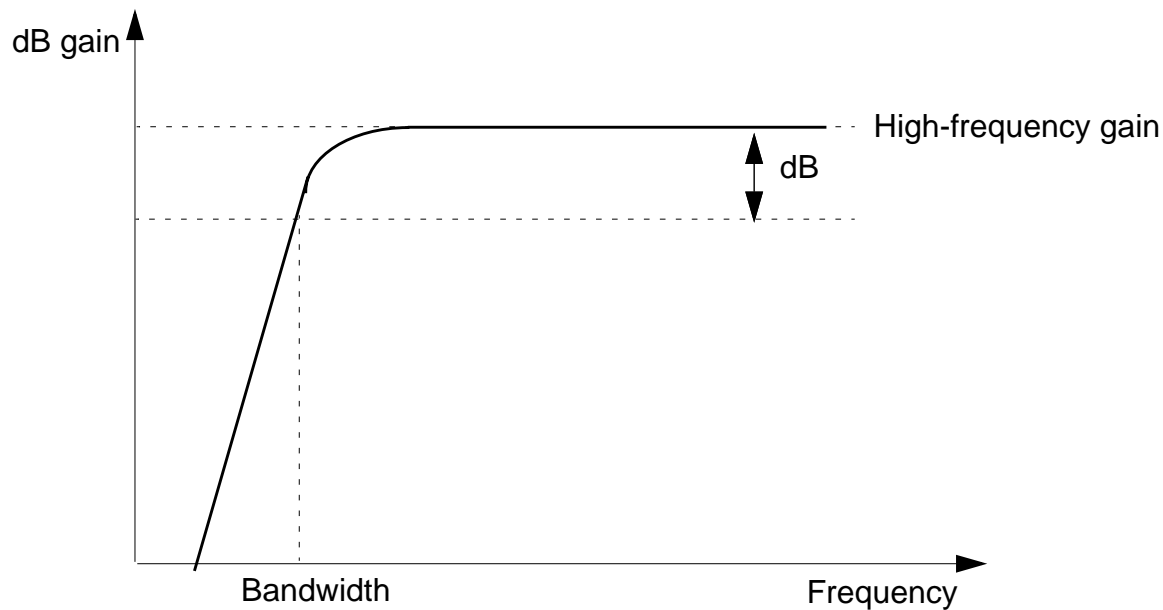
## Waveform Calculator User Guide

### Algebraic Mode

---

frequency in the response waveform. The *dB* field specifies *n*. An error occurs if the magnitude of the input waveform does not drop *n* decibels below the gain at high frequency.

#### High-Pass Bandwidth Function



#### Computing Band-Pass Bandwidth

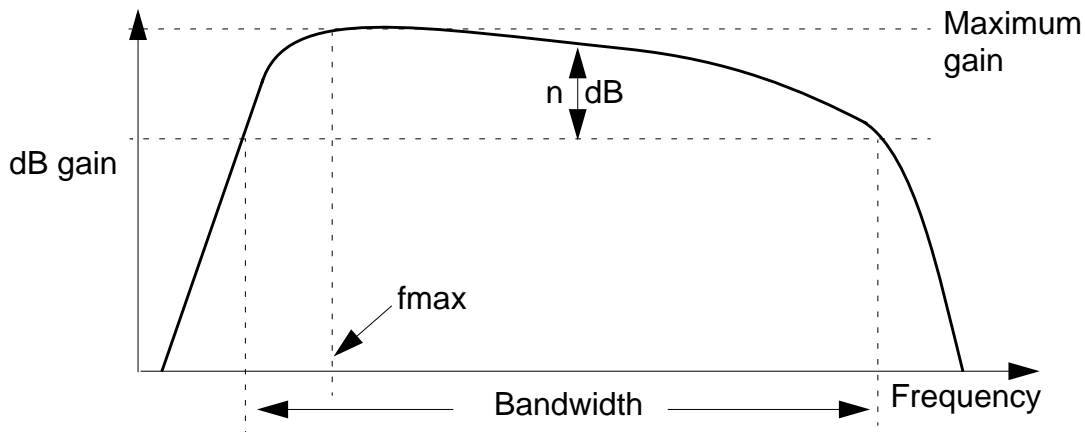
The calculator computes the band-pass bandwidth by

1. Determining the lowest frequency ( $f_{\max}$ ) at which the magnitude of the input waveform is maximized
2. Determining the highest frequency less than  $f_{\max}$  at which the input waveform magnitude drops *n* decibels below the maximum (*n* is the number you enter in the *dB* field)
3. Determining the lowest frequency greater than  $f_{\max}$  at which the input waveform magnitude drops *n* decibels below the maximum

## Waveform Calculator User Guide

### Algebraic Mode

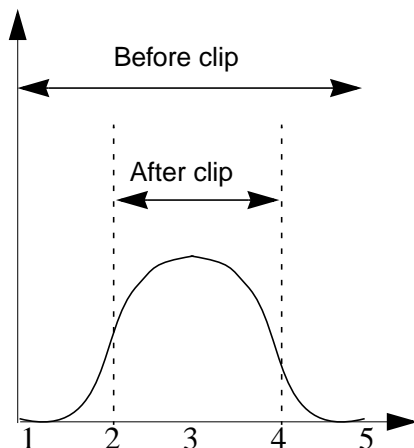
4. Subtracting the value returned by step 2 from the value returned by step 3. The value returned by step 2 or step 3 must exist.



## Clip Function

The *clip* function restricts the waveform defined by the buffer expression to the range entered in the *From* and *To* fields. You can use the *clip* function to restrict the range of action of other special functions of the calculator such as [integ](#), [rms](#), and [frequency](#).

1. Set up the expression you want to clip in the buffer.
2. Select *clip*.
3. Click *Get Buffer*.
4. Enter the limits and click *OK*.



Clip			
OK	Cancel	Defaults	Apply
Get Buffer	<input type="text"/>		
From	<input type="text" value="2"/>	To	<input type="text" value="4"/>

## Compression Function

This function returns the  $Nth$  compression point value of a waveform at the extrapolation point that you specify. To use this function:

1. Set up the ne600p mixer cell from `dfII/samples/artist/rfExamples` library. The design variable `frf` should be set to 920MHz.
2. Ensure that the sourcetype on the rf port is 'sine' and the Amplitude (dBm) is set to `prf`.
3. Set up a PSS analysis. The beat frequency should be set to 40MHz. Set the number of harmonics to 2 (only two harmonics are required to determine the 1 dB compression point). Sweep the `prf` parameter from -30 to 10 in 10 linear steps.
4. Set the Model Library path to include the `dfII/samples/artist/models/spectre/rfModels.scs` file.

5. After running the simulation, call up the Waveform Calculator and the Results Browser. Click on *schematic->psf->Run1->sweep\_pss\_pss\_fd-sweep->sweepVariable->prf->10->sweep\_pss-004\_pss-fd.pss->Pif* with the left mouse button. The following will appear in the calculator buffer:

```
v( "/Pif" ?result "sweep_pss_pss_fd-sweep" ?resultsDir "~/simulation/ne600p/spectre/schematic" ).
```

6. Click on *Special Functions -> compression*. The *Compression* form will be displayed.

Compression				
OK	Cancel	Defaults	Apply	Help
Get Buffer	<input type="text"/>			
Harmonic Num.	<input type="text" value="2"/>	Ext. Point (X)	<input type="text" value="-25"/>	Compression dB
	<input type="text" value="1"/>			<input type="text" value="1"/>

7. Click *Get Buffer* in the *Compression* form.
8. Enter *Harmonic number*=2. This is the second harmonic of the 40 MHz fundamental frequency, which is the IF frequency (80MHz).
9. Enter *Ext. Point (X)* = -25 field to specify the extrapolation point of the waveform. The extrapolation point is the X axis value.
10. Enter *Compression dB* = 1 to specify the compression coefficient (N).
11. Click on the *OK* button.

## Waveform Calculator User Guide

### Algebraic Mode

---

12. Click the *Evaluate Buffer* button in the Calculator. The result appears in the Calculator display.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## CompressionVRI Function

This function performs an Nth compression point measurement on a power waveform.

Use this function to simplify the declaration of a compression measurement. This function extracts the specified harmonic from the input waveform(s), and uses `dBm(spectralPower((i or v/r),v))` to calculate a power waveform. The function then passes this power curve and the remaining arguments to the *compression* function to complete the measurement.

The *compression* function uses the power waveform to extrapolate a line of constant slope (dB/dB) according to a specified input or output power level. This line represents constant small-signal power gain (ideal gain). The function then finds the point where the power waveform drops N dB from the constant slope line and returns either the x coordinate (input referred) or y coordinate (output referred) value.

To use this function:

1. Define the voltage waveform in the buffer.
2. Choose *compressionVRI* in the *Special Functions* menu.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Compression form opens.

CompressionVRI

OK Cancel Defaults Apply Help

Get Buffer

Harmonic  Extrapolation Point

Load Resistance  Compression dB

Input Referred Compression ☐

3. Click *Get Buffer* in the CompressionVRI form.
4. Type a value in the *Harmonic* field to specify the harmonic index of the waveform.
5. Type a value in the *Extrapolation Point* field to specify the extrapolation point for the waveform. The default value is the minimum x value of the input voltage waveform.

The extrapolation point is the coordinate value in dBm that indicates the point on the output power waveform where the constant-slope power line begins. This point should be in the linear region of operation.
6. Type a numerical value in the *Load Resistance* field. The default value is 50.
7. In the *Compression dB* field, type the delta (in dB) between the power waveform and the ideal gain line that marks the compression point. The default value is 1.
8. Choose whether the measurement is for *Input Referred Compression* or *Output Referred Compression*.
9. Click *OK*.

## Convolution (Convolve) Function

The *convolve* function computes the convolution of two waveforms.

1. In the *Special Functions* menu of the calculator, choose *convolve*.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Convolution form appears.

Convolution				
OK	Cancel	Defaults	Apply	
Get Buffer				
Get Buffer				
From		To		linear <input type="checkbox"/>
			By	

2. In the Convolution form, define the first waveform in the upper buffer and click the upper *Get Buffer* button.
3. Define the second waveform in the lower buffer and click the lower *Get Buffer* button.
4. Specify the range, scale, and increment by values and click *OK*.

Convolution is defined as

$$\int_{from}^{to} f1(s)f2(t-s)ds$$

$f1$  and  $f2$  are the functions defined by the first and second waveforms.

**Note:** The *convolve* function is numerically intensive and might take longer than the other functions to compute.

### Threshold Crossing (cross) Function

The threshold crossing or *cross* function computes the x-axis value *xcross* at which the *n*th crossing of the specified edge type of the threshold value occurs.

1. Select *cross*.

## Waveform Calculator User Guide

### Algebraic Mode

The Threshold Crossing form appears.

Threshold Crossing			
OK	Cancel	Defaults	Apply
Help			
Get Buffer			
Threshold Value	2.5	Edge Number	1
		Edge Type	either <input type="checkbox"/>

2. Set up the expression in the calculator buffer.

3. Click *Get Buffer* in the Threshold Crossing form.

The expression is copied to the Threshold Crossing form buffer.

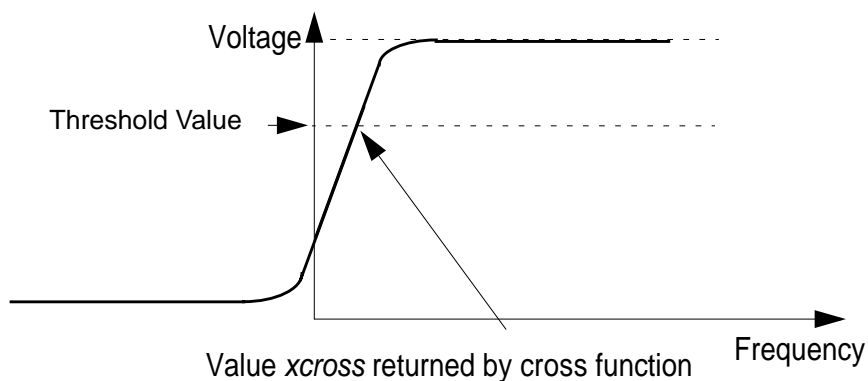
4. Enter the threshold value of the waveform at which to perform the calculation.

5. In *Edge Number*, enter the number of the crossing at which to perform the calculation.

The integer you enter specifies which crossing is returned. (For example, 1 specifies the first crossing, and 2 specifies the second crossing.)

If you specify a positive integer, the count starts at the smallest  $x$  value of the waveform, and the search is in the direction of increasing  $x$  values. If you specify a negative integer, the count starts at the largest  $x$  value of the waveform, and the search is in the direction of decreasing  $x$  values. If you enter 0, all the crossings found are returned in a list.

6. Select an *Edge Type* to determine the crossing as the rising edge, falling edge, or either edge.



7. Click *OK*.

## dBm Function

The *dBm* function performs the operation

$$\text{dB10}(x) + 30$$

1. Select *dBm*.
2. Enter the value for  $x$  and close the parentheses.

## Delay Function

The *delay* function computes the delay between two points using the [cross function](#).

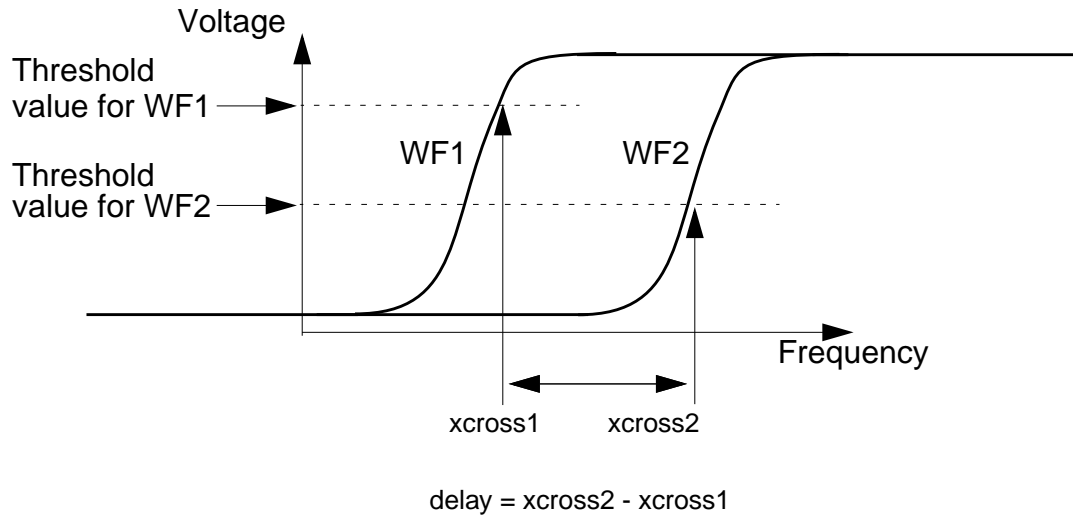
1. Select *delay*.

The Threshold Delay form appears.

Threshold Delay				
<b>OK</b>	<b>Cancel</b>	<b>Defaults</b>	<b>Apply</b>	<b>Help</b>
Wf1:	<input type="button" value="Get Buffer"/>	<div style="border: 1px solid black; height: 20px;"></div>		
Threshold Value	<input style="width: 80px;" type="text" value="2.5"/>	Edge Number	<input style="width: 80px;" type="text" value="1"/>	Edge Type <input style="width: 80px;" type="text" value="either"/>
Wf2:	<input type="button" value="Get Buffer"/>	<div style="border: 1px solid black; height: 20px;"></div>		
Threshold Value	<input style="width: 80px;" type="text" value="2.5"/>	Edge Number	<input style="width: 80px;" type="text" value="1"/>	Edge Type <input style="width: 80px;" type="text" value="either"/>

2. Enter the first waveform into the upper buffer and click *Wf1: Get Buffer*.
3. Enter the second waveform into the lower buffer and click *Wf2: Get Buffer*.

4. Specify the threshold values, edge numbers, and edge types and click *OK*.



## Derivative (deriv) Function

The *deriv* function computes the derivative of an expression. You can plot the resulting waveform.

1. Select *deriv*.
2. Enter the expression and closing parenthesis into the calculator buffer.

**Note:** After the second derivative, the results become inaccurate because the derivative is obtained numerically.

## Discrete Fourier Transform (dft) Function

The tool which converts a temporal (time domain) description of a signal (real or complex) into one, in terms of its frequency components is called the Fourier transform. DFT (Discrete Fourier Transform) is the discrete formulation of the Fourier transform, which takes such regularly spaced data values (samples in time domain), and returns the value of the Fourier transform for a set of values in frequency domain which are equally spaced. Most of the time, however, we work on real-valued signals only.

Consider a complex series (signal)  $w(k)$  with  $N$  samples of the form

## Waveform Calculator User Guide

### Algebraic Mode

---

$w(0), w(1), w(2), \dots, w(k), \dots, w(N-1)$

Further, assume that the series outside the range  $0, N-1$  is extended  $N$ -periodic, that is,  $w(k) = w(k+N)$  for all  $k$ . The DFT of this series will be denoted  $W(n)$ , will also have  $N$  samples and will be defined as:

$$W(n) = \frac{1}{N} \sum_{k=0}^{N-1} w(k) \left( e^{-2\pi i k \frac{n}{N}} \right) \quad \text{where } n = 0, \dots, N-1$$

#### Note:

- The first sample  $W(0)$  of the transformed series is the DC component, more commonly known as the average of the input series.
- The DFT of a real series results in a symmetric series about the Nyquist frequency (described below).
- The highest positive (or negative) frequency sample is called the Nyquist frequency. This is the highest frequency component that should exist in the input series for the DFT to receive 'unpredictable' results. More specifically, if there are no frequencies above Nyquist frequency, the original signal can be exactly reconstructed from the samples. The Nyquist Theorem (or Shannon's Sampling Theorem) exactly specifies this, that for a band limited signal, you must sample at a frequency over twice the maximum frequency of the signal, to reconstruct it from the samples.

While the DFT transform above can be applied to any complex valued series, in practice for large series it can take considerable time to compute, the time taken being proportional to the square of the number of points (samples) in the series. A much faster algorithm has been developed by Cooley and Tukey called the FFT (Fast Fourier Transform). The only requirement of the most popular implementation of this algorithm (Radix-2 Cooley-Tukey) is that the number of points in the series be a power of 2 i.e.  $N=2^n$ .

Given  $N$  input points, the FFT returns  $N$  frequency components, of which the first  $(N/2 + 1)$  are valid. (The other components are mirror images and are considered invalid since the frequencies they represent do not satisfy the Nyquist Theorem above.) They start with the DC component, and are spaced apart by a frequency of  $(1 / (n \text{ delta } T))$ . The magnitude of the complex number returned is the frequency's relative strength.

## Waveform Calculator User Guide

### Algebraic Mode

---

The *dft* function computes the discrete Fourier transform of the buffer by FFT algorithm where  $\text{delta}T = (t2 - t1) / N$ . The waveform is sampled at the following  $N$  timepoints:

$t1, t1 + \text{delta}T, t1 + 2 * \text{delta}T, \dots, t1 + (N - 1) * \text{delta}T$

The output of *dft()* is a frequency waveform,  $W(f)$ , which has  $(N/2 + 1)$  complex values: the dc term, the fundamental, and  $(N/2 - 1)$  harmonics.

**Note:** The last time point,  $(t1 + (N - 1) * \text{delta}T)$ , is  $(t2 - \text{delta}T)$  rather than  $t2$ . The *dft* function assumes that  $w(t1)$  equals  $w(t2)$ . To use the *dft* function

1. Select *dft*.

The Discrete Fourier Transform form appears.

The screenshot shows a dialog box titled "Discrete Fourier Transform". It has a standard Windows-style interface with a title bar, a menu bar (containing a minus sign), and a toolbar with buttons for "OK", "Cancel", "Defaults", "Apply", and "Help". The main area of the dialog contains several controls: a "Get Buffer" button followed by a text input field; a "From" label with a spin box, a "To" label with a spin box, and a "Number of Samples" label with a text input field containing the value "64"; a "Window Type" label with a dropdown menu showing "Rectangular" and a small square icon; a "Smoothing Factor" label with a text input field containing the value "1"; and a "Coherent Gain" label with a dropdown menu showing "(none)" and a small square icon, followed by a text input field containing the value "1".

2. Specify the range over which you want to compute the transform.

Be sure to cover at least one complete period of your slowest frequency.

3. Enter the number of samples you want to take in expanding the Fourier transform.

This number should be a power of 2. If it is not, the system increases the value to the next higher power of 2. Sample at a rate that is at least twice your highest frequency component (the Nyquist rate). Pick a sampling rate high enough that closely spaced frequency components can be resolved.

4. Select the *Window Type* option.

For more information, see the [table](#) of window type option values later in this section.

5. Specify the *Smoothing Factor* (for the Kaiser window type only).

The *Smoothing Factor* field accepts values from 0 to 15. The value 0 implies no smoothing and is equivalent to a rectangular window. The default value for the *Smoothing Factor* field is 1.

6. Click OK.

## Waveform Calculator User Guide

### Algebraic Mode

---

When you run the transient analysis, keep the maximum time step small enough to represent the highest frequency component accurately. The maximum time step should be smaller than the sampling period that you use for the discrete Fourier transform (DFT) of the time domain waveform. The samples in the DFT will either hit a data point (calculated exactly by the simulator) or an interpolated point between two data points.

Choosing a maximum time step during transient simulation that is smaller than the DFT sampling period ensures that sampling maintains a resolution at least equal to that of the transient time-domain waveform.

The start and stop times should not coincide with the boundaries of the time-domain waveform. The boundary solutions might be imprecise and generate incorrect results if used in other calculations.

One of the uses of fast Fourier transform (FFT) windowing is to reduce discontinuities at window edges caused by having a nonintegral number of periods of a signal in a window. This removes the abrupt edges, making them fall off smoothly to zero, and can improve the validity of the FFT components obtained. You can also use FFT windowing to 'dig out' the details of signal components that are very close in frequency or that consist of both large and small amplitudes.

The following table was obtained from the book *The FFT, Fundamentals and Concepts* by R. W. Ramirez, Prentice Hall, 1985. As explained in this reference, the values in the table were computed from software-generated windows and might vary slightly from theoretical values. In the third column, the peak magnitude of each window is compared with that of the rectangular window. In the fourth column, the amplitude of the highest side lobe is given in decibels referenced to the major lobe peak. The fifth column contains the 3dB bandwidth of the major lobe, normalized to one over the window's width. The last column gives the theoretical rolloff of the side lobes.

Window name	Shape equation	Major lobe height	Highest side lobe (dB)	Band-width (3 dB)	Theor. rolloff (dB/octave)
Cosine4	$A = (.5(1 - \cos(2\pi t/T)))^2$ for $t=0$ to $T$	$0.36T$	-46.9	$1.79/T$	30
ExtCosBell	$A = 0.5(1 - \cos(2\pi 5t/T))$ for $t=0$ to $T/10$ and $t=9T/10$ to $T$ $A=1$ for $t=T/10$ to $9T/10$	$0.9T$	-13.5	$0.95/T$	18 (beyond $5/T$ )

## Waveform Calculator User Guide

### Algebraic Mode

Window name	Shape equation	Major lobe height	Highest side lobe (dB)	Bandwidth (3 dB)	Theor. rolloff (dB/octave)
HalfCycleSine	$A = \sin(2\pi 0.5t/T)$ for $t=0$ to $T$	$0.64T$	-22.4	$1.15/T$	12
HalfCycleSine3	$A = \sin^3(2\pi 0.5t/T)$ for $t=0$ to $T$	$0.42T$	-39.5	$1.61/T$	24
HalfCycleSine6					
Hamming	$A = .08 + .46(1 - \cos(2\pi t/T))$ for $t = 0$ to $T$	$0.54T$	-41.9	$1.26/T$	6 (beyond $5/T$ )
Kaiser					
Parzen	$A = 1 - 6(2t/T - 1)^2 + 6 2t/T - 1 ^3$ for $t=T/4$ to $3T/4$ $A = 2(1 -  2t/T - 1 )^3$ for $t=0$ to $T/4$ and $t=3T/4$ to $T$	$0.37T$	-53.2	$1.81/T$	24
Rectangular	$A = 1$ for $t=0$ to $T$	$T$	-13.2	$0.86/T$	6

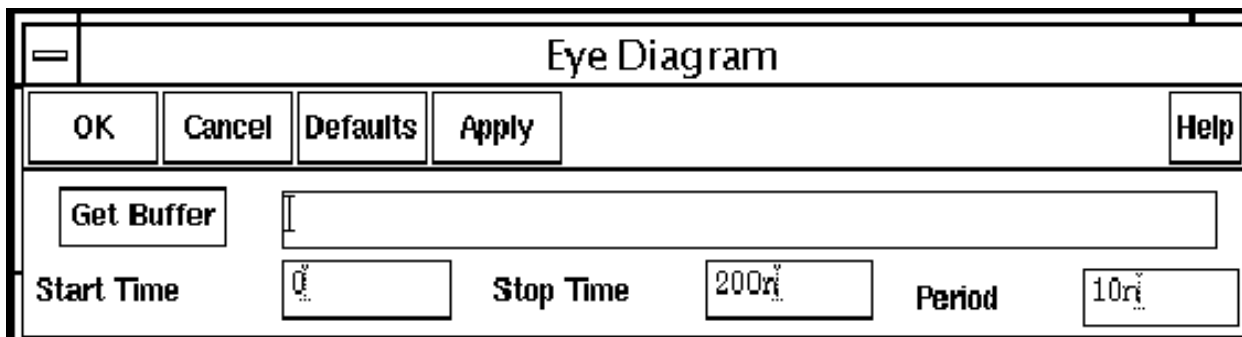
### Sources of Errors

- *dft()* performs interpolation to determine values of  $w(t)$  that are not directly available from the simulator output. This interpolation can cause an inaccurate spectrum.
- If  $(t_2 - t_1)$  is not the time period of  $w(t)$ , the output of *dft()* might be misleading.
- If the simulator-generated values are inaccurate, *dft()* returns a frequency waveform with many 'insignificant' harmonics. You can minimize the amplitudes of these harmonics by increasing the accuracy of the simulator.

## eyeDiagram Function

The *eyeDiagram Function* gives an eye-diagram plot in which the waveform signal is divided into fixed time periods, which are then superimposed on each other. The result is a plot that has many overlapping lines enclosing an empty space known as the "eye". The quality of the receiver circuit is characterized by the dimension of the eye. An open eye means that the detector will be able to distinguish between 1's and 0's in its input, while a closed eye means that a detector placed on  $V_{out}$  is likely to give errors for certain input bit sequences.

1. Choose *eyeDiagram* from the *Special Functions* menu in the *Calculator*. The Eye Diagram form appears.



The screenshot shows a dialog box titled "Eye Diagram". It contains a row of buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below these buttons is a "Get Buffer" button and a text input field. At the bottom of the dialog, there are three labeled input fields: "Start Time" with a small icon, "Stop Time" with the value "200ns", and "Period" with the value "10ns".

2. Define a waveform in the buffer.
3. Enter the values for Start Time, Stop Time and Period.
4. Click OK.
5. Click on the *plot button* in the *Calculator* to plot the eyeDiagram.

## Flip Function

The *flip* function negates the X values. It has the effect of flipping the waveform horizontally in relation to the y axis.

## Fourier Evaluation (fourEval) Function

The *fourEval* function evaluates the Fourier series represented by an expression. This function is an inverse Fourier transformation and thus the inverse of the Discrete Fourier Transform (dft) Function. The *fourEval* function transforms the expression from the frequency domain to the time domain.

1. Select *fourEval*.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Fourier Evaluation form appears.

Fourier Evaluation				
OK	Cancel	Defaults	Apply	Help
Get Buffer				
From	1	To	20	By 2

2. Set up the expression in the calculator buffer and click *Get Buffer*.
3. Specify the time range over which you want to evaluate the series.
4. Enter the increment for evaluating the series.

## Frequency Function

The *frequency* function estimates the frequency of a periodic waveform. The system computes the reciprocal of the average time between two successive midpoint crossings of the rising waveform.

1. Select *frequency*.
2. Enter the expression and the closing parenthesis.

## Gain (gainBwProd/gainMargin) Functions

The *gainBwProd* function calculates the gain-bandwidth product. This function requires one argument, the frequency response of interest over a sufficiently large frequency range.

$$\text{gainBwProd}(\text{gain}) = A_o * f2$$

The gain-bandwidth product is calculated as the product of the DC gain  $A_0$  and the critical frequency  $f2$ . The critical frequency  $f2$  is the smallest frequency for which the gain equals  $1/\sqrt{2}$  times the DC gain  $A_0$ .

The *gainMargin* function computes the gain margin of the loop gain of an amplifier. It requires one argument, the loop gain of interest over a sufficiently large frequency range.

$$\text{gainMargin}(\text{gain}) = 20 * \log_{10}(\text{value}(\text{gain } f0))$$

## Waveform Calculator User Guide

### Algebraic Mode

---

The gain margin is calculated as the magnitude of the gain in dB at  $f_0$ . The frequency  $f_0$  is the smallest frequency in which the phase of the gain provided is -180 degrees. For stability, the gain margin must be less than 0dB.

1. Select *gainBwProd* or *gainMargin*.
2. Enter the expression and the closing parenthesis.

## Group Delay Function

The *groupDelay* function computes the group delay of an expression.

Group delay is defined as the derivative of the phase with respect to frequency. Group delay is expressed in seconds. It is calculated using the *vp* function as shown below.

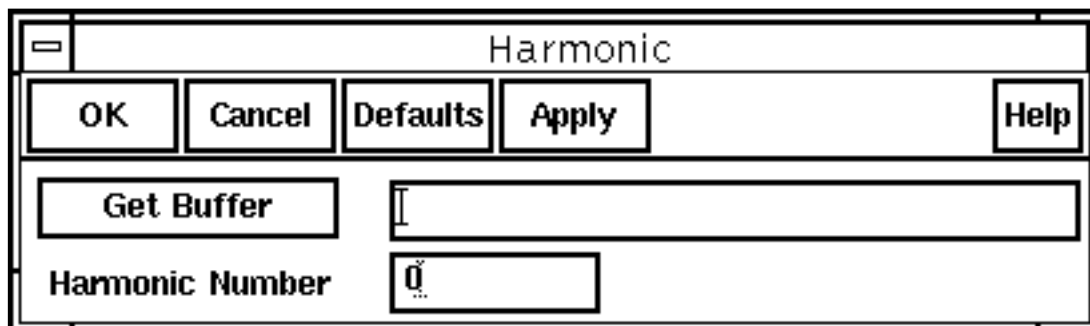
$$\text{Group Delay} = \frac{d\phi}{d\omega} = \frac{d}{df} \left[ \frac{\text{phase}(/netX)}{360} \right]$$

1. Select *groupDelay*.
2. Enter the expression and the closing parenthesis.

## Harmonic Function

This function returns the harmonic waveform of a waveform you specify.

1. Choose *harmonic* in the *Special Functions* menu of the Calculator to open the Harmonic form.



The screenshot shows a dialog box titled "Harmonic". It contains a row of buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below these buttons is a "Get Buffer" button and a text input field. Below the text input field is a label "Harmonic Number" and a spin box showing the value "0".

2. Enter the expression for the waveform in the Calculator buffer.

## Waveform Calculator User Guide

### Algebraic Mode

---

3. Click *Get Buffer* in the Harmonic form.
4. In the Harmonic form, specify the harmonic you want by typing the value in the *Harmonic Number* field.
5. Click *OK* or *Apply* in the Harmonic form.
6. Click *plot* in the Calculator to plot the waveform.

To use this function, you must type the line below in the CIW

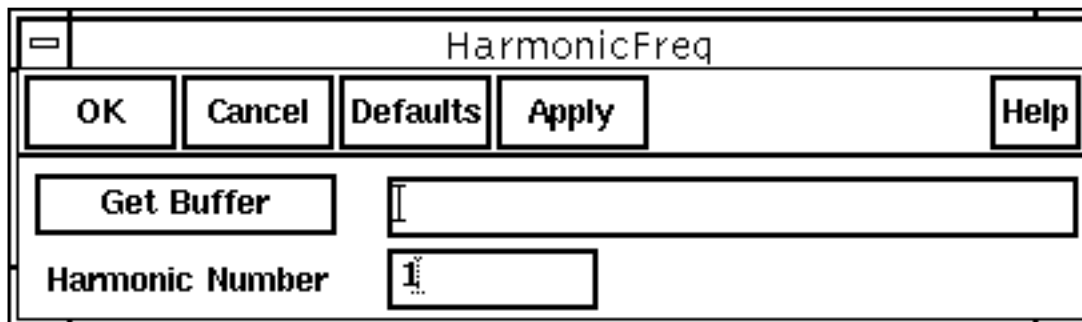
```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

### Harmonic Frequency Function

This function returns the harmonic waveform of a waveform you specify.

1. Enter the expression for the waveform in the Calculator buffer.
2. Choose *harmonicFreq* in the *Special Functions* menu of the Calculator to open the HarmonicFreq form.



The image shows a dialog box titled "HarmonicFreq". At the top, there is a row of buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below this row, there is a "Get Buffer" button followed by a text input field. Below that, there is a label "Harmonic Number" followed by a text input field containing the number "1".

3. Click *Get Buffer* in the HarmonicFreq form.
4. In the HarmonicFreq form, specify the harmonic you want by typing the value in the *Harmonic Number* field.
5. Click *OK* or *Apply*.

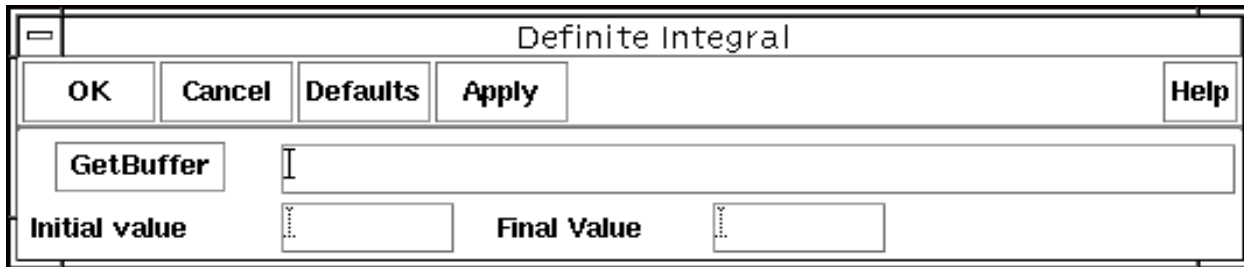
### iinteg Function

The *iinteg* function computes the indefinite integral of an expression with respect to the X-axis variable. The result is a waveform that can be plotted.

## integ Function

The *integ* function computes the definite integral of the expression in the buffer. The result is the value of the area under the curve over a specified range on the X-axis of the expression.

1. Choose *integ* from the *Special Functions* menu in Calculator. The Definite Integral form appears.



The image shows a software dialog box titled "Definite Integral". It features a standard Windows-style title bar with a close button. Below the title bar are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". The main area of the dialog contains a "GetBuffer" button on the left and a large text input field on the right. At the bottom of the dialog, there are two input fields labeled "Initial value" and "Final Value".

2. Define a waveform in the buffer.
3. Enter the values for the limits of the definite integral in the *Initial Value* and *Final Value* fields.

**Note:** You should specify either both the limits or neither. In case you do specify the limits, they become the end points of the range on the X-axis for definite integration. If you do not specify the limits, then the range for definite integration is the entire range of the sweep on the X-axis.

4. Click *OK*.

## ipn Function

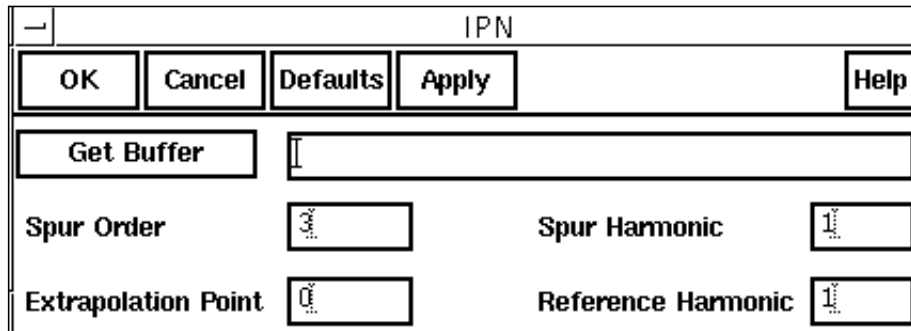
This function plots the *Nth* order intercept between two harmonics of a waveform that you define.

## Waveform Calculator User Guide

### Algebraic Mode

---

1. Choose *ipn* from the *Special Functions* menu in the Calculator. The IPN form opens.



IPN				
OK	Cancel	Defaults	Apply	Help
Get Buffer	<input type="text"/>			
Spur Order	<input type="text" value="3"/>	Spur Harmonic	<input type="text" value="1"/>	
Extrapolation Point	<input type="text" value="0"/>	Reference Harmonic	<input type="text" value="1"/>	

2. Enter the expression for the waveform in the Calculator buffer.
3. Click *Get Buffer* in the IPN form.
4. Type values for the following four quantities:

Spur Order	Spur Order determines what order of interference is calculated for the spurious and reference waves. The default value is 3; this corresponds to the <i>IP3</i> function. If you use a value other than 3, that order of interference is calculated between those two waves.
Spur Harmonic	Harmonic number for spurious waveform.
Reference Harmonic	Harmonic number for reference waveform.
Extrapolation Point	The extrapolation point for the <i>ipn</i> function. This is the X axis value.

5. Click *OK* or *Apply*.

The expression is sent to the calculator buffer. To evaluate the expression, click the equal sign (=) button in the calculator.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## ipnVRI Function

This function performs an intermodulation Nth-order intercept point measurement.

## Waveform Calculator User Guide

### Algebraic Mode

Use this function to simplify the declaration of an ipn measurement. This function extracts the spurious and reference harmonics from the input waveform(s), and uses `dBm(spectralPower((i or v/r),v))` to calculate the respective powers. The function then passes these power curves or numbers and the remaining arguments to the *ipn* function to complete the measurement.

From each of the spurious and reference power waveforms (or points), the *ipn* function extrapolates a line of constant slope (dB/dB) according to the specified order and input power level. These lines represent constant small-signal power gain (ideal gain). The *ipn* function calculates the intersection of these two lines and returns the value of either the x coordinate (input referred) or y coordinate.

1. Choose *ipnVRI* in the *Special Functions* menu of the Calculator. The ipnVRI form opens.

The image shows a software dialog box titled "ipnVRI". At the top, there are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below these buttons is a "Get Buffer" button followed by a large text input field. The main body of the dialog contains several labeled input fields: "Spur Harmonic", "Reference Harmonic", "Spur Order", "Extrapolation Point", and "Load Resistance". To the right of these is a checkbox labeled "Input Referred IPN". At the bottom, there is a label "Circuit Input Power is:" followed by a checkbox labeled "Variable Sweep".

2. Enter the expression for the waveform in the Calculator buffer.
3. Click *Get Buffer* in the ipnVRI form.
4. Type values for the following quantities:
  - Spur Harmonic* Harmonic index for spurious waveform.
  - Reference Harmonic* Harmonic index for reference waveform.

## Waveform Calculator User Guide

### Algebraic Mode

---

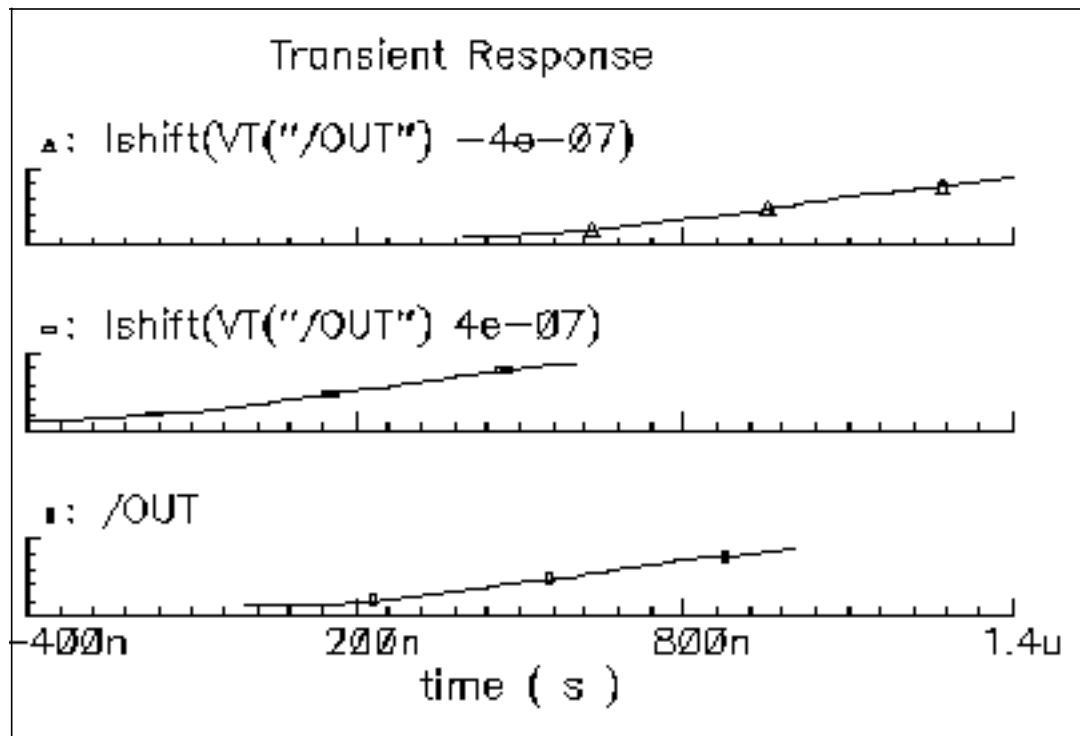
<i>Spur Order</i>	Spur Order determines what order of interference is calculated for the spurious and reference waves. The default value is 3; this corresponds to the <i>IP3</i> function. If you use a value other than 3, that order of interference is calculated between those two waves.
<i>Extrapolation Point</i>	The extrapolation point for the <i>ipn</i> function. This is the X axis value. The default is the minimum x value of the input voltage waveform.
<i>Load Resistance</i>	The resistance into the output port. The default value is 50.

5. To get the X-coordinate of the intercept, specify *Input Referred IPN*. To get the Y-coordinate of the intercept, specify *Output Referred IPN*.
6. Indicate whether the *Circuit Input Power* is a *Variable Sweep* or a *Single Point*.
7. Click *OK* or *Apply*.

**Note:** For an extended design example, using *ipn* and *ipnVRI* functions, see [Appendix A, "Using the Calculator Special Functions with SpectreRF Simulation Results"](#).

## Lshift Function

The *Lshift* function shifts the data in the Waveform window to the left by a specified amount. A negative value shifts the data to the right.



## Minimum and Maximum Functions

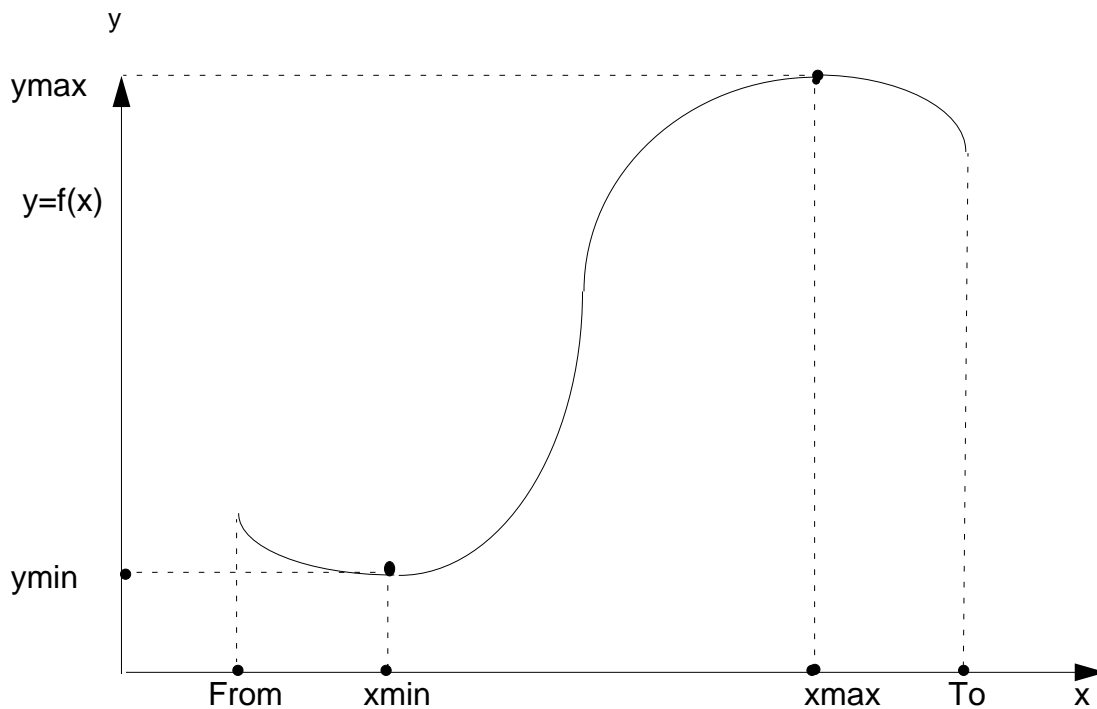
You can calculate minimum and maximum values of waveforms with the *xmax*, *xmin*, *ymin*, and *ymin* functions.

This figure shows the relationship of these functions.

## Waveform Calculator User Guide

### Algebraic Mode

---



### **$xmax$ and $ymin$**

The  $xmax$  function computes the value of the independent variable  $x$  at which the expression attains its maximum value, that is, the value of  $x$  that maximizes  $y=f(x)$ .

1. Select  $xmax$ .

The X Value at Maximum form appears.

X Value at Maximum				
OK	Cancel	Defaults	Apply	Help
Get Buffer				
Nth Maximizer	1			

2. Enter the expression in the calculator buffer and click *Get Buffer*.

## Waveform Calculator User Guide

### Algebraic Mode

---

The maximum might occur at more than one point on the x axis. You must choose (in the *Nth Maximizer* field) which maximum value you want to see. The calculator returns the value of the Nth Maximizer counting from the left, that is, toward increasing X-axis values. If you enter a negative integer, the direction of search is reversed toward decreasing X-axis values (counting from the right).

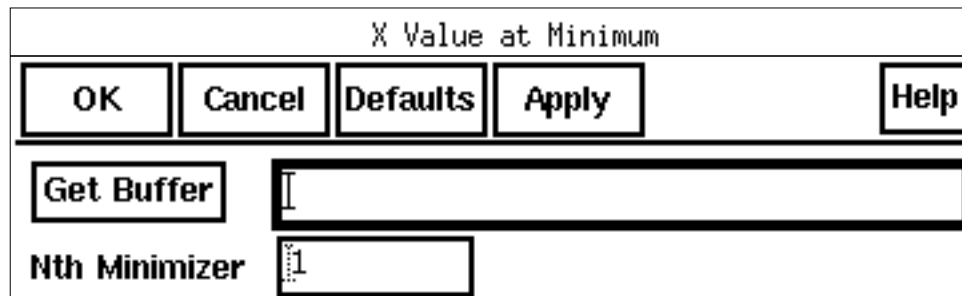
The *y*max function computes the maximum y of the expression  $y=f(x)$ .

#### xmin and ymin

The *x*min function computes the value of the independent variable x at which the expression has its minimum value, that is, the value of x that minimizes  $y=f(x)$ .

1. Select *x*min.

The X Value at Minimum form appears.



X Value at Minimum	
OK	Cancel Defaults Apply Help
Get Buffer	<input type="text"/>
Nth Minimizer	<input type="text" value="1"/>

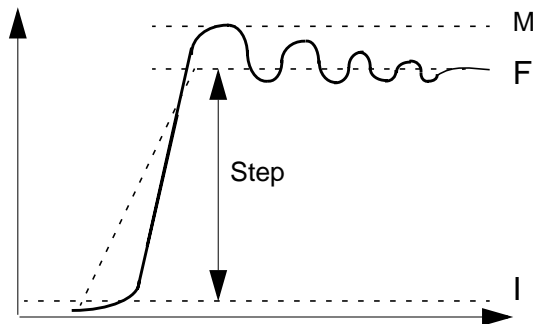
2. Enter the expression in the calculator buffer and click *Get Buffer*.

The minimum might occur at more than one point on the x axis. You must choose (in the *Nth Minimizer* field) which minimum value you want to see. The calculator returns the value of the Nth Minimizer, counting from the left, that is, toward increasing X-axis values. If you enter a negative integer, the direction of search is reversed toward decreasing X-axis values (counting from the right).

The *y*min function computes the minimum y of the expression  $y=f(x)$ .

## Overshoot Function

The *overshoot* function computes the percentage by which an expression overshoots a step going from the Initial Value to the Final Value you enter.



$$\text{Overshoot} = \frac{M - F}{F - I}$$

1. Select *overshoot*.

Overshoot				
<b>OK</b>	<b>Cancel</b>	<b>Defaults</b>	<b>Apply</b>	<b>Help</b>
<b>Get Buffer</b>	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>			
Initial Value	◆ y at x	◇ y	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>	
Final Value	◆ y at x	◇ y	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>	

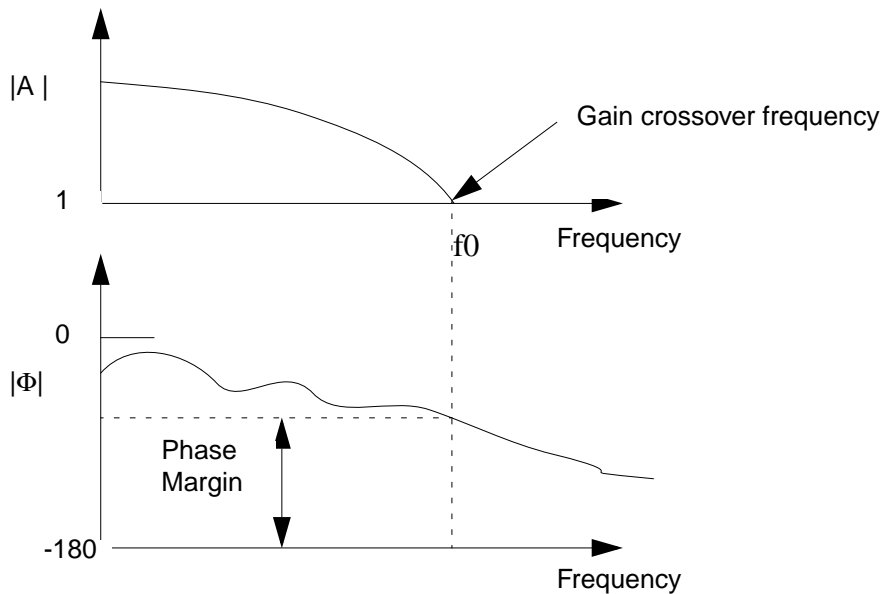
2. Enter the expression in the calculator buffer and click *Get Buffer*.
3. To set the *Initial* and *Final Values*, do the following:
  - ☐ Use the *y* option and enter the *y* values directly (for example, 3V) or a horizontal [marker](#) name (for example, M2).
  - ☐ Use the *y at x* option to enter an *x* value or vertical marker name (for example, M1).

The system calculates the corresponding *y* value for the current waveform.

## Phase Margin Function

The *phaseMargin* function computes the phase margin of the loop gain of an amplifier. This function is similar to the [phaseDegUnwrapped](#) Waveform Calculator SKILL command. It requires one argument, the loop gain of interest over a sufficiently large frequency range.

```
phaseMargin( gain ) = 180 + phase( value( gain f0 ) )
```



The phase margin is calculated as the difference between the phase of the gain in degrees at  $f_0$  and at -180 degrees. The frequency  $f_0$  is the smallest frequency where the gain is 1. For stability, the phase margin must be positive.

1. Select *phaseMargin*.
2. Enter the expression and the closing parenthesis.

## Phase Noise Function

This function plots the phase noise waveform for noise analysis results. You need to follow the following steps to use this function.

1. Set up your PSS analysis.
2. Check the *oscillator button* in the *choosing analysis* form.
3. Set up a Pnoise analysis. Note the value you are entering for the relative harmonic in the *choosing analysis* form.

## Waveform Calculator User Guide

### Algebraic Mode

---

4. Run your simulation.
5. Invoke the Calculator . Choose *phaseNoise* in the *Special Functions* menu of the Calculator. The *Select Analysis* form appears.

Name	Type	Description
------	------	-------------

6. Choose the pnoise-pnoise analysis name in the *Analysis Name* field of the *Select Analysis* form, or select the analysis name from the list box in the form. Click on the *OK* button in the form. The *Phase Noise* form appears.

Harmonic: 1      Noise Result Name: pnoise.pss

7. In the Phase Noise form, enter *harmonic number*=1 or the value you had entered for relative harmonic in the *pnoise choosing analysis* form.
8. Enter the *Noise Result Name* as pnoise-pnoise and click on *OK* in the form.
9. In your calculator buffer, you should see something like `phaseNoise(1, "pss-fd.pss", "result "pnoise-pnoise")`.
10. Click on the *plot* button in the Calculator to plot the waveform.
11. The phase noise plot will appear.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" `boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Power Spectral Density (psd) Function

The power spectral density (*psd*) function describes how the power (or variance) of a time series (signal) is distributed with frequency. Mathematically, it is defined as the Fourier Transform of the auto correlation sequence of the time series (signal). The waveform is first interpolated, to generate evenly spaced data points in time. The spacing of the data points is the inverse of the *dft* sampling frequency. The *psd* is computed by first breaking up the time interval into overlapping segments. Each segment is multiplied, time point by time point, by the specified windowing function. The *dft* is performed on each windowed segment of the baseband waveform. At each frequency, the *dfts* from all segments are averaged together and the squared modulus of these averages, gives *psd*.

After you choose *psd* in the *Special Functions* menu of the Calculator, the Power Spectral Density form opens.

Power Spectral Density

OK Cancel Defaults Apply Help

Get Buffer [ ]

From [ ] To [ ] Number of Samples 512

Window Type Hanning [ ] Smooth. Fac. 1 Window Size 256

Coherent Gain (none) [ ] 1 Detrending Mode None [ ]

1. Enter the expression for the waveform in the Calculator buffer, and then click *Get Buffer*.
2. In the *From* field, type the starting time for the spectral analysis interval.
3. In the *To* field, type the ending time for the spectral analysis interval. You can use this parameter and the *From* parameter to exclude part of the interval. For example, you might set these values to discard initial transient data.
4. In the *Number of Samples* field, type the number of time domain points to use. The maximum frequency in the Fourier analysis is proportional to the *Number of Samples* parameter and inversely proportional to the difference between the starting time and the ending time.
5. Choose the *Window Type* that you want to use. If you select the Kaiser window type, then type in a value for the Kaiser smoothing factor. The smoothing factor must be in the range  $0 \leq \text{factor} \leq 15$ , where 0 is the same as using a rectangular window.
6. In the *Window Size* field, type in the number of frequency domain points to use in the Fourier analysis.

## Waveform Calculator User Guide

### Algebraic Mode

A larger window size results in an expectation operation over fewer samples, which leads to larger variations in the power spectral density. A small window size can smear out sharp steps in the power spectral density that might really be present.

7. Choose a *Coherent Gain* factor. If you choose *magnitude*, *dB20*, or *dB10*, then enter a scaling factor. A non-zero factor scales the power spectral density by  $1/(\text{factor})$ . Valid values for the factor are  $0 < \text{factor} < 1$ . You can also use a value of 1 if you do not want the *Coherent Gain* factor to be used.

8. Choose which *Detrending Mode* to use.

The *psd* function works by applying a moving windowed FFT to time-series data. If there is a deterministic trend to the underlying data, you might want to remove the trend before performing the spectral analysis. For example, consider analyzing phase noise in a VCO model. Without the noise the phase increases more or less linearly with time, so it is appropriate to set the detrending mode to *linear*. To subtract an average value, set the detrending mode to *mean*. Where the spectrum of raw data is desired, set the detrending mode to *none*.

9. Click *OK*.

## Power Spectral Density Baseband (psdbb) Function

The power spectral density baseband (*psdbb*) function returns an estimate for the power spectral density of a waveform1+j \* waveform2.

After you choose *psdbb* in the *Special Functions* menu of the Calculator, the Power Spectral Density Baseband form opens.

The screenshot shows a dialog box titled "Power Spectral Density Baseband". It has a standard Windows-style interface with buttons for "OK", "Cancel", "Defaults", "Apply", and "Help". The dialog contains several input fields and controls:

- Two "Get Buffer" buttons, each followed by a text input field.
- "From" and "To" text input fields.
- "Number of Samples" text input field with the value "512".
- "Window Type" dropdown menu set to "Hanning", with a "Smooth. Fac." text input field set to "1".
- "Window Size" text input field set to "256".
- "Coherent Gain" dropdown menu set to "(none)", with a text input field set to "1".
- "Detrending Mode" dropdown menu set to "None".

1. Enter the expression for the waveform in the Calculator buffer, and then click *Get Buffer*.

## Waveform Calculator User Guide

### Algebraic Mode

---

2. In the *From* field, type the starting time for the spectral analysis interval.
3. In the *To* field, type the ending time for the spectral analysis interval. You can use this parameter and the *From* parameter to exclude part of the interval. For example, you might set these values to discard initial transient data.
4. In the *Number of Samples* field, type the number of time domain points to use. The maximum frequency in the Fourier analysis is proportional to the *Number of Samples* parameter and inversely proportional to the difference between the starting time and the ending time.
5. Choose the *Window Type* that you want to use. If you select the *Kaiser* window type, then type in a value for the Kaiser smoothing factor. The smoothing factor must be in the range  $0 \leq \text{factor} \leq 15$ , where 0 is the same as using a rectangular window.
6. In the *Window Size* field, type in the number of frequency domain points to use in the Fourier analysis.

A larger window size results in an expectation operation over fewer samples, which leads to larger variations in the power spectral density. A small window size can smear out sharp steps in the power spectral density that might really be present.

7. Choose a *Coherent Gain* factor. If you choose *magnitude*, *dB20*, or *dB10*, then enter a scaling factor. A non-zero factor scales the power spectral density by  $1/(\text{factor})$ . Valid values for the factor are  $0 < \text{factor} < 1$ . You can also use a value of 1 if you do not want the *Coherent Gain* factor to be used.
8. Choose which *Detrending Mode* to use.

The *psddb* function works by applying a moving windowed FFT to time-series data. If there is a deterministic trend to the underlying data, you might want to remove the trend before performing the spectral analysis. For example, consider analyzing phase noise in a VCO model. Without the noise the phase increases more or less linearly with time, so it is appropriate to set the detrending mode to *linear*. To subtract an average value, set the detrending mode to *mean*. Where the spectrum of raw data is desired, set the detrending mode to *none*.

9. Click *OK*.

## Rise Time Function

The *riseTime* function computes the rise time of an expression, that is, the time required to change from the value in the *Percent Low* field to the value in the *Percent High* field of the difference between *Initial Value* and *Final Value*.

## Waveform Calculator User Guide

### Algebraic Mode

1. Select *riseTime*.

The Rise Time form appears.

Rise Time

OK Cancel Defaults Apply Help

Get Buffer

Initial Value ☒ y at x ☐ y

Final Value ☒ y at x ☐ y

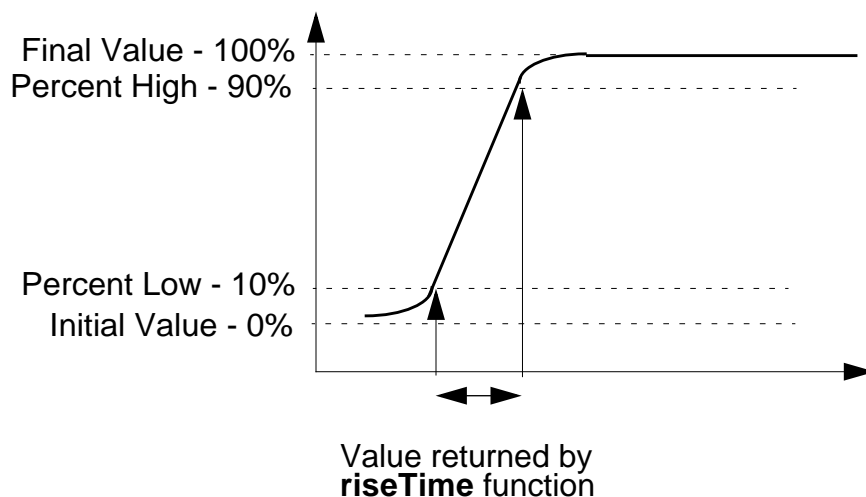
Percent Low 10 Percent High 90

2. Set up an expression in the calculator buffer and click *Get Buffer*.

3. To set the initial and final values in the Rise Time form you can:

- Use the *y* option for "Initial Value" and "Final Value" and enter the *y* values directly or a horizontal [marker](#) name.
- Use the *y at x* option for "Initial Value" and "Final Value" to enter an *x* value or vertical [marker](#) name.

4. Click *OK*.



## Waveform Calculator User Guide

### Algebraic Mode

---

The above graph represents the “Initial value” of the signal as 0% and “Final value” as 100%. The “Percent Low” and “Percent High ” values are taken as 10% and 90%.

For waveforms with multiple rise and fall edges, it is suggested that users should isolate edges of interest by using the clip() function or enter values for y at x rather than entering y values directly in the Rise Time function input form.

### Root-Mean-Square (rms) Function

The *rms* function computes the root-mean-square value of the expression *f(x)*, over the specified range of *x*. This is the square root of the integral of the expression squared over the specified range, divided by the range.

For example, if  $y=f(x)$ ,

$$rms(y) = \sqrt{\frac{\int_{from}^{to} f(x)^2 dx}{to - from}}$$

1. Select *rms*.
2. Enter the expression and closing parenthesis.

To compute the rms value of the expression over a smaller range, use the [clip function](#) inside the *rms* function.

### Root-Mean-Square (rms) Noise Function

The *rmsNoise* function computes the integrated root-mean-square noise over the bandwidth specified in hertz in the *From* and *To* fields.

RMS Noise				
OK	Cancel	Defaults	Apply	Help
From	<input type="text"/>	To	<input type="text"/>	

## Waveform Calculator User Guide

### Algebraic Mode

---

**Note:** To instead plot the squared noise voltage versus frequency, use the [Results – Plot Noise command](#) in the simulation window.

## Root Function

The *root* function computes the value of  $x$  at which  $f(x)$  equals the specified threshold.

1. Select *root*.

The Root Value form appears.

The screenshot shows a dialog box titled "Root Value". It has a title bar with the text "Root Value". Below the title bar, there are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Below these buttons, there is a "Get Buffer" button on the left and a large empty text input field on the right. Below the "Get Buffer" button, there are two input fields: "Threshold Value" with the value "2.5" and "Nth Root" with the value "1".

2. Set up an expression in the calculator buffer and click *Get Buffer*.
3. Enter the waveform value at which to compute the root value.
4. Enter the *root* you want to see.

## Sample Function

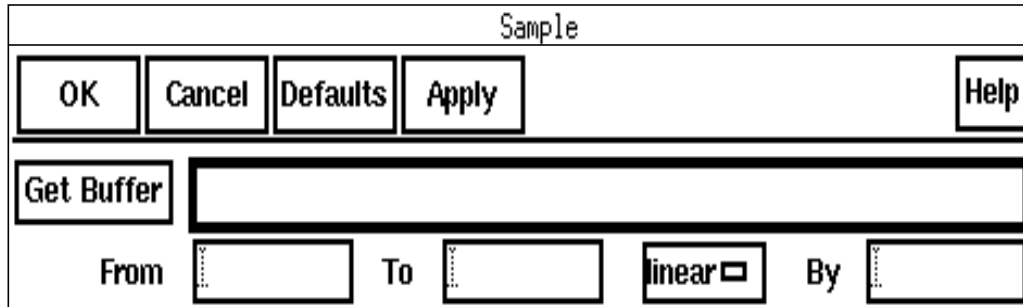
The *sample* function samples a waveform at the interval you specify. You can use this function to speed up plotting of waveforms that have many data points.

1. Select *sample*.

## Waveform Calculator User Guide

### Algebraic Mode

The Sample form appears.



The 'Sample' form contains a title bar 'Sample' and a row of buttons: 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'. Below this is a 'Get Buffer' button and a large text input field. At the bottom, there are labels 'From' and 'To' followed by input fields, a 'linear' checkbox, and a 'By' label followed by an input field.

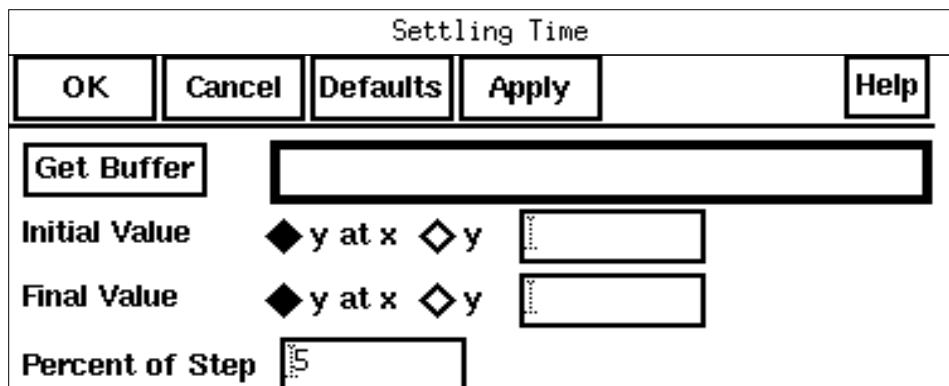
2. Define a waveform in the calculator buffer.
3. Specify the range and increment.

If you sample a waveform beyond its range, you get the final value of the waveform. You can use this function to demodulate a signal. Consider an AM modulated sine wave. Assume the carrier frequency is 1 GHz, and the modulation frequency is 1 MHz. If the waveform is sampled every 1 ns, the resulting signal will be cleanly demodulated (the 1 GHz carrier is completely eliminated by the sampling).

## Settling Time Function

The *SettlingTime* is the time by which the signal settles within the specified "Percent of step" of the difference between the "Final Value" and "Initial Value" from the "Final Value".

1. Select *settlingTime*. The Settling Time form appears.



The 'Settling Time' form has a title bar 'Settling Time' and buttons 'OK', 'Cancel', 'Defaults', 'Apply', and 'Help'. It includes a 'Get Buffer' button and a large text input field. Below these are two rows for 'Initial Value' and 'Final Value', each with a 'y at x' selection (a diamond icon) and a 'y' selection (a diamond icon), followed by a 'Percent' input field. At the bottom, there is a 'Percent of Step' label and an input field with the value '5'.

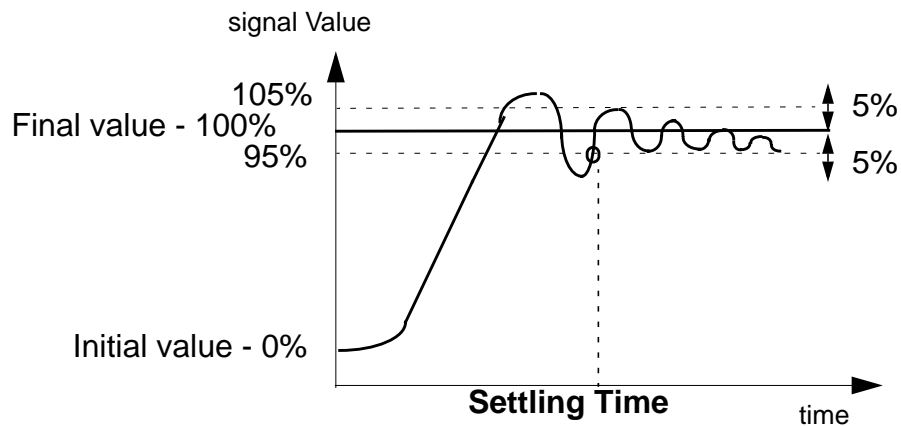
2. Set up an expression in the calculator buffer and click *Get Buffer*.
3. To set the initial and final values in the Settling Time form you can:

## Waveform Calculator User Guide

### Algebraic Mode

---

- Use the *y* option for "Initial Value" and "Final Value" and enter the *y* values directly or a horizontal [marker](#) name.
  - Use the *y at x* option for "Initial Value" and "Final Value" to enter an *x* value or vertical [marker](#) name.
4. Enter the value for *Percent of Step*.
  5. Click *OK*.



The above graph represents the "Initial value" of the signal as 0% and "Final value" as 100%. The "Percent of Step" is taken as 5%.

## Slew Rate Function

The *slewRate* function computes the average rate at which an expression changes from percent low to percent high of the difference between initial value and final value.

1. Select *slewRate*.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Slew Rate form appears.

Slew Rate

OK Cancel Defaults Apply Help

Get Buffer

Initial Value ☒ y at x ☐ y

Final Value ☒ y at x ☐ y

Percent Low 10 Percent High 90

2. Set up an expression in the calculator buffer and click *Get Buffer*.
3. To set the initial and final values, do one of the following:
  - Use the *y* option for "Initial Value" and "Final Value" and enter the *y* values directly or a horizontal [marker](#) name.
  - Use the *y at x* option for "Initial Value" and "Final Value" to enter an *x* value or vertical [marker](#) name.
4. Click *OK*.

For waveforms, with multiple rise and fall edges, it is recommended that users isolate edges of interest by using the `clip()` function or enter values for *y at x* rather than entering *y* values directly in the Slew Rate function input form.

## Spectral Power Function

This function plots the spectral power for a current waveform and a voltage waveform that you define. To use this function:

1. Choose *spectralPower* in the *Special Functions* menu of the Calculator.
2. Define the voltage waveform.
3. Click the *space* button on the Calculator.
4. Define the current waveform.
5. Click the right parenthesis button “)” on the Calculator to close the expression.

6. Click *plot* in the Calculator to plot the waveform.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Standard Deviation (*stddev*) Function

The *stddev* function computes the standard deviation of a waveform (or a family of waveforms) over its entire range. Standard deviation (*stddev*) is defined as the square-root of the variance where variance is the integral of the square of the difference of the expression  $f(x)$  from average ( $f(x)$ ), divided by the range of  $x$ .

For example, if  $y=f(x)$

$$stddev(y) = \sqrt{\frac{\int_{from}^{to} (y - average(y))^2}{to - from}}$$

If you want a different range, use the [clip function](#) to clip the waveform to the range you want.

## Table Function

The *table* function defines a piecewise linear function from a column of  $x$  and  $y$  values in a file. This function was previously named *implicitX*.

1. Select *table*.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Table form appears.

Table			
OK	Cancel	Defaults	Apply
File Name	<input type="text"/>		
Function Name	<input type="text"/>		
X Column Number	<input type="text" value="1"/>	Y Column Number	<input type="text" value="2"/>
X Skip Lines	<input type="text" value="0"/>	Y Skip Lines	<input type="text" value="0"/>

2. Enter the name of the data file and any name for the function.
3. (Optional) Enter the column numbers containing the X- and Y-axis data, if they are not in columns 1 and 2 respectively.
4. (Optional) Enter the number of lines to skip in each column from the top of the file before reading the data.

**Note:** Do not count comment lines beginning with a semicolon and blank lines in the number of lines to skip.

The X data must be real numbers increasing monotonically. The Y data can be real numbers, or complex numbers following this syntax:

```
(real_part imag_part)
```

or

```
complex(real_part imag_part)
```

## Tangent Function

This function plots a line that passes through x and y coordinates and the slope that you specify. To use this function:

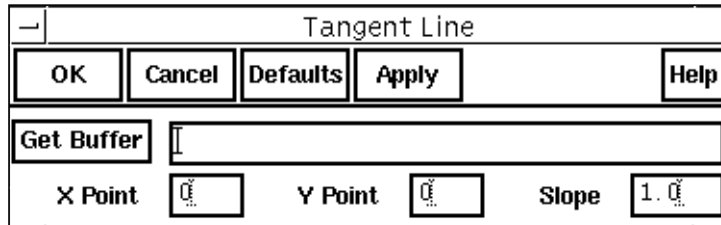
1. Choose *tangent* from the *Special Functions* menu in the Calculator.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Tangent Line form appears.



Tangent Line				
OK	Cancel	Defaults	Apply	Help
Get Buffer	<input type="text"/>			
X Point	<input type="text" value="0.0"/>	Y Point	<input type="text" value="0.0"/>	Slope <input type="text" value="1.0"/>

2. Define a waveform in the buffer.
3. Click *Get Buffer* in the Tangent Line form.
4. Type values for the *X point*, *Y Point*, and *Slope* fields.
5. Click *OK*.
6. Click *plot* in the Calculator to plot the tangent line.

To use this function, you must type the line below in the CIW

```
envSetVal("calculator" "oldexpr" 'boolean nil)
```

or set the calculator `oldexpr` variable to `nil` in your `.cdsenv` file.

## Total Harmonic Distortion (thd) Function

The *thd* function computes the percentage of total harmonic content of a signal with respect to the fundamental frequency. The computation uses the *dft* function (for information, see [Discrete Fourier Transform \(dft\) Function](#)). Assume that the *dft* function returns complex coefficients  $A_0, A_1, \dots, A_f, \dots$ . Please note that fundamental frequency  $f$  is the frequency contributing to the largest power in the signal.  $A_0$  is the complex coefficient for the DC component and  $A_i$  is the complex coefficient for the  $i$ th harmonic where  $i \neq 0, f$ . Then, total harmonic distortion is computed as:

$$\frac{\sqrt{\sum_{i=1, i \neq 0, f} |A_i|^2}}{|A_f|} \times 100 \%$$

To compute the *thd*, you need to perform the following steps:

1. Choose *thd* in the *Special Functions* menu of the calculator.

## Waveform Calculator User Guide

### Algebraic Mode

---

The Total Harmonic Distortion form appears.

The screenshot shows a dialog box titled "Total Harmonic Distortion". At the top are buttons for "OK", "Cancel", "Defaults", "Apply", and "Help". Below these is a "Get Buffer" button and a text input field. Underneath are "From" and "To" time input fields, a "Number of Samples" input field with the value "64", and a "Fundamental (Hz)" input field. A note at the bottom says "Enter 0 to choose the largest signal".

2. Set up an expression in the calculator buffer and click *Get Buffer*.
3. Specify the range and the number of samples.
4. Click *OK*.
5. Click *Print* to see the result.

The accuracy of the total harmonic distortion measurement depends on simulator options and the analysis parameters. For an accurate measurement set the following simulation options:

Option	Suggested Value
RELTOL	1e-5
ABSTOL	1e-13
VNTOL	3e-8
TRTOL	1
METHOD	gear
MAXORD	3

Set the simulation timestep to be 1/100th of a cycle, and simulate for ten cycles. End the simulation slightly beyond the tenth cycle. When you use the calculator, measure during the tenth cycle by specifying the beginning of the cycle as the From time and the end as the To time.

## Value Function

The *value* function computes the value of the waveform at the point you specify.

1. Select *value*.

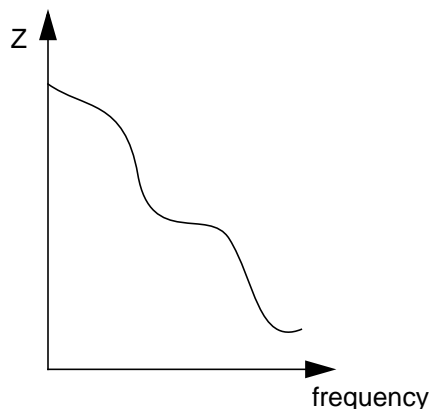
The Value form appears.

Value	
OK	Cancel
Defaults	Apply
<div style="border: 1px solid black; padding: 2px; display: inline-block;">Help</div>	
<div style="border: 1px solid black; padding: 2px; display: inline-block;">Get Buffer</div>	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>
Interpolate At	<div style="border: 1px solid black; height: 20px; width: 100%;"></div>

2. Set up an expression in the calculator buffer and click *Get Buffer*.
3. Enter the point at which to compute the value and click *OK*.

## X Value (xval) Function

The X value function (*xval*) takes a single expression as an argument. It returns a waveform with the Y values equal to the X values:  $Y=X$ . This facilitates computations where the dependent variable (such as time or frequency) is needed in an expression. For example, you can use *xval* to compute the capacitance waveform for this curve:



$$\text{Capacitance Waveform } C(f) = \frac{1}{2\pi Z \cdot xval(Z)}$$

Here are some syntax examples:

## Waveform Calculator User Guide

### Algebraic Mode

---

`xval(VF("/IN"))`

`xval(VT("/net3"))`

1. Select *xval*.
2. Enter the expression and the closing parenthesis.

---

## RF Mode

The waveform calculator supports two sets of buttons or modes: standard and RF. To switch between the two modes, click the *RF* or *standard* button at the top right of the calculator. You can switch repeatedly between the RF and standard modes, because switching does not affect a calculation in progress.

In the RF mode, the waveform calculator provides mathematical functions used for RF circuit design. In this mode the keypad includes the following:

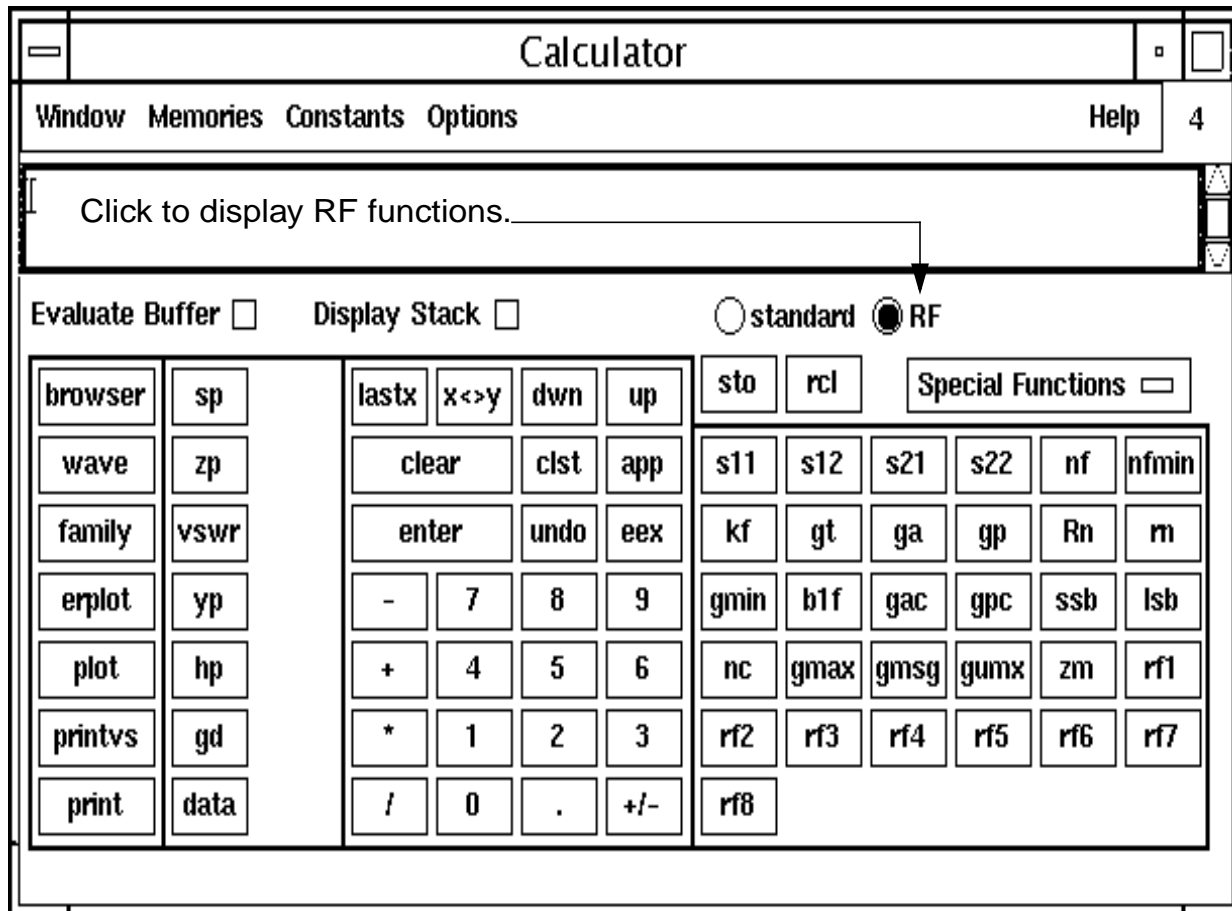
- The same numerical keypad, special functions, and results output buttons as the standard mode.
- Two banks of function buttons for the RF function. Each bank includes functions for linear S-parameter results and for harmonic balance results.  
  
**Note:** The RF calculator function is expected to work with SP analysis. So, for these functions to work, you need to run an SP analysis along with any other chosen analyses.
- User-definable function buttons.

Many of the RF button functions are also available on the [S-Parameters Results form](#) for direct plotting.

## Waveform Calculator User Guide

### RF Mode

Click *RF* to open the RF version of the calculator.



Click in the picture of the calculator for descriptions of the RF calculator functions, or continue reading here for a general description of the calculator.

The RF mode calculator buttons are used for two types of analyses: S-parameter and harmonic balance. This section describes only the functions that are used with the RF mode. You can also see [descriptions](#) and [mathematical expressions](#) for the functions associated with each calculator key.

**Dual port functions** relate one quantity to another. An example is  $SP(2\ 1)$ , which expresses the reflected wave at port 2 in terms of the incident wave at port 1. The *s11* button yields the expression  $SP(1\ 1)$ , and the expression is complete. This approach is limited to two ports because the number of *s##* buttons equals the upper limit of the number of ports squared. For more ports, use the *sp* function. Other dual port functions are *zp*, *yp*, and *hp*.

To enter the *sp* expression (or any other dual port expression):

## Waveform Calculator User Guide

### RF Mode

---

1. Click on the *sp* button. The prompt “Select the reflected port” appears at the bottom of the CIW and the design window.
2. In the schematic, click on a *psin* element from *analogLib*. This is the reflected (signal) port. The function does not let you select another type of object. The prompts change to “Select the incident port.”
3. Click on the incident (reference) port. The complete two-port expression appears in the calculator buffer.

**Single port functions** involve only one port. The only single port function on the calculator is *vswr*. To use *vswr*:

1. Click on the *vswr* button. The prompt “Select the port for expression VSWR...” appears at the bottom of the CIW and the design window.
2. In the schematic, click on one of the *psin* elements of *analogLib*. The function does not let you select another type of object. The expression appears in the buffer.

**Constant performance circles** include *gac*, *gpc*, *ssb*, *lsb*, and *nc*. Use these functions only when the Waveform window is set to display a Smith chart.

### Programmable Buttons

The *rf#* buttons along the bottom row of the right bank of calculator buttons are user function buttons. You can program them to perform whatever function you want, just as you can program the *f#* functions on the standard bank of buttons. Refer to “Assigning Function Keys” on page 41 for information about programming user function buttons.

### Data Button

## Waveform Calculator User Guide

### RF Mode

---

When you click on the *data* button, the Select Analysis form opens up, listing all of the analyses already performed on the current design.

Name	Type	Description
pss-td.pss	td.pss	
pss-fd.pss	fd.pss	
variables	design_variables	

Click on the analysis in the list that you want to review.

The design window and CIW prompt you to “Select net, terminals, or instance for the DATA expression.” Click on the name of an analysis to select it. You can use this function to quickly review various analyses that you have just performed.

## Waveform Calculator User Guide

### RF Mode

---

#### Calculator Buttons for RF

---

Button	Result	Arguments
<b>For Linear S-Parameter Results:</b>		
<i>sp</i>	Scattering parameters	Signal & reference port #s
<i>yp</i>	Admittance parameters	Signal & reference port #s
<i>zp</i>	Impedance parameters	Signal & reference port #s
<i>hp</i>	H-parameters	Signal & reference port #s
<i>gd</i>	Group delay	Signal & reference port #s
<i>data</i>	Plots a previous analysis	Net or terminal name and analysis name
<i>gmsg</i>	Maximum stable power gain for a two port.	Signal & reference port #s
<i>gmax</i>	Maximum available gain for a two port	Signal & reference port #s
<i>gumx</i>	Maximum unilateral power gain for a two port	Signal & reference port #s
<i>zm</i>	Input impedance if all other ports are matched	

# Waveform Calculator User Guide

## RF Mode

Button	Result	Arguments
<i>vswr</i>	Voltage standing wave ratio	A port number
<i>s11, s12, s21, s22</i>	2-port S-parameters	None
<i>nf*</i>	Noise factor For more information about this function, please refer <b>Note</b> given at the end of this table.	$\Gamma_s$ (the source reflection coefficient) can be a single complex value or a waveform. Default is 0 (50 ohms).
<i>nfmin*</i>	Minimum noise factor For more information about this function, please refer <b>Note</b> given at the end of this table.	None
<i>kf</i>	Stability factor K	None
<i>gt</i>	Transducer gain	$\Gamma_s$ and $\Gamma_L$ (source & load reflection coefficients). Each can be a single complex value or a waveform. Default is 0 (50 Ohms).
<i>ga</i>	Available gain	$\Gamma_s$ (source reflection coefficient) can be a single complex value or a waveform. Default is 0 (50 Ohms).
<i>gp</i>	Power gain	$\Gamma_L$ (load reflection coefficient) can be a single complex value or a waveform. Default is 0 (50 Ohms).
<i>Rn</i>	Equivalent noise resistance	None
<i>rn</i>	Normalized equivalent noise resistance	None
<i>gmin</i>	Optimum noise reflection coefficient for NFmin	None
<i>b1f</i>	Stability factor b1f	None
<i>gac</i>	Available gain circles	Gain circle value in dB
<i>gpc</i>	Power gain circles	Gain circle value in dB
<i>ssb</i>	Source stability circles	None

## Waveform Calculator User Guide

### RF Mode

Button	Result	Arguments
<i>lsb</i>	Load stability circles	None
<i>nc</i>	Noise circles	Noise figure value in dB

**Note:** *nf* and *nfmin* buttons in Calculator refer to noise factors. These buttons simply retrieve the *F* and *Fmin* respectively from the psf file. In the psf file, *NF* (Noise Figure), *NFmin* (Minimum Noise Figure), *F* (Noise Factor), and *Fmin* (Minimum Noise Factor) exist. They are related to each other by the following set of equations.

$$NF = db10(F)$$

$$NFmin = db10(Fmin)$$

The user can compute the values for noise figure, by taking the *db10* value of the corresponding noise factor data, using the *db10* Calculator button.

## Special Function Buttons for RF

Clicking *Special Functions* in the calculator opens a menu of functions that you can use in RF analyses. For more information, follow the cross-references in the following table.

Function	For RPN mode, see	For algebraic mode, see
<i>average</i>	<a href="#">“Average Function” on page 52</a>	<a href="#">“Average Function” on page 99</a>
<i>bandwidth</i>	<a href="#">“Bandwidth Function” on page 53</a>	<a href="#">“Bandwidth Function” on page 99</a>
<i>clip</i>	<a href="#">“Clip Function” on page 56</a>	<a href="#">“Clip Function” on page 102</a>
<i>compression</i>	<a href="#">“Compression Function” on page 56</a>	<a href="#">“Compression Function” on page 103</a>
<i>compressionVRI</i>	<a href="#">“CompressionVRI Function” on page 58</a>	<a href="#">“CompressionVRI Function” on page 104</a>
<i>convolve</i>	<a href="#">“Convolution (convolve) Function” on page 59</a>	<a href="#">“Convolution (Convolve) Function” on page 105</a>
<i>cross</i>	<a href="#">“Threshold Crossing (cross) Function” on page 60</a>	<a href="#">“Threshold Crossing (cross) Function” on page 106</a>
<i>dBm</i>	<a href="#">“dBm Function” on page 61</a>	<a href="#">“dBm Function” on page 108</a>
<i>delay</i>	<a href="#">“Delay Function” on page 61</a>	<a href="#">“Delay Function” on page 108</a>

# Waveform Calculator User Guide

## RF Mode

Function	For RPN mode, see	For algebraic mode, see
<i>deriv</i>	<a href="#">“Derivative (deriv) Function”</a> on page 62	<a href="#">“Derivative (deriv) Function”</a> on page 109
<i>dft</i>	<a href="#">“Discrete Fourier Transform (dft) Function”</a> on page 62	<a href="#">“Discrete Fourier Transform (dft) Function”</a> on page 109
<i>eyeDiagram</i>	<a href="#">eyeDiagram Function</a> on page 66	<a href="#">eyeDiagram Function</a> on page 114
<i>flip</i>	<a href="#">“Flip Function”</a> on page 67	<a href="#">“Flip Function”</a> on page 114
<i>fourEval</i>	<a href="#">“Fourier Evaluation (fourEval) Function”</a> on page 67	<a href="#">“Fourier Evaluation (fourEval) Function”</a> on page 114
<i>frequency</i>	<a href="#">“Frequency Function”</a> on page 68	<a href="#">“Frequency Function”</a> on page 115
<i>gainBwProd</i>	<a href="#">“Gain (gainBwProd/gainMargin) Functions”</a> on page 68	<a href="#">“Gain (gainBwProd/gainMargin) Functions”</a> on page 115
<i>gainMargin</i>	<a href="#">“Gain (gainBwProd/gainMargin) Functions”</a> on page 68	<a href="#">“Gain (gainBwProd/gainMargin) Functions”</a> on page 115
<i>groupDelay</i>	<a href="#">“Group Delay Function”</a> on page 68	<a href="#">“Group Delay Function”</a> on page 116
<i>harmonic</i>	<a href="#">“Harmonic Function”</a> on page 68	<a href="#">“Harmonic Function”</a> on page 116
<i>harmonicFreq</i>	<a href="#">“Harmonic Frequency Function”</a> on page 69	<a href="#">“Harmonic Frequency Function”</a> on page 117
<i>iinteg</i>	<a href="#">“iinteg Function”</a> on page 70	<a href="#">“iinteg Function”</a> on page 117
<i>integ</i>	<a href="#">“integ Function”</a> on page 70	<a href="#">“integ Function”</a> on page 118
<i>ipn</i>	<a href="#">“ipn Function”</a> on page 70	<a href="#">“ipn Function”</a> on page 118
<i>ipnVRI</i>	<a href="#">“ipnVRI Function”</a> on page 71	<a href="#">“ipnVRI Function”</a> on page 119
<i>lshift</i>	<a href="#">“Lshift Function”</a> on page 73	<a href="#">“Lshift Function”</a> on page 122
<i>overshoot</i>	<a href="#">“Overshoot Function”</a> on page 76	<a href="#">“Overshoot Function”</a> on page 125
<i>phaseMargin</i>	<a href="#">“Phase Margin Function”</a> on page 77	<a href="#">“Phase Margin Function”</a> on page 126
<i>phaseNoise</i>	<a href="#">“Phase Noise Function”</a> on page 77	<a href="#">“Phase Noise Function”</a> on page 126

# Waveform Calculator User Guide

## RF Mode

Function	For RPN mode, see	For algebraic mode, see
<i>psd</i>	<a href="#">“Power Spectral Density (psd) Function” on page 79</a>	<a href="#">“Power Spectral Density (psd) Function” on page 128</a>
<i>psddb</i>	<a href="#">“Power Spectral Density Baseband (psddb) Function” on page 80</a>	<a href="#">“Power Spectral Density Baseband (psddb) Function” on page 129</a>
<i>riseTime</i>	<a href="#">“Rise Time Function” on page 82</a>	<a href="#">“Rise Time Function” on page 130</a>
<i>rms</i>	<a href="#">“Root-Mean-Square (rms) Function” on page 83</a>	<a href="#">“Root-Mean-Square (rms) Function” on page 132</a>
<i>rmsNoise</i>	<a href="#">“Root-Mean-Square (rms) Noise Function” on page 83</a>	<a href="#">“Root-Mean-Square (rms) Noise Function” on page 132</a>
<i>root</i>	<a href="#">“Root Function” on page 83</a>	<a href="#">“Root Function” on page 133</a>
<i>sample</i>	<a href="#">“Sample Function” on page 84</a>	<a href="#">“Sample Function” on page 133</a>
<i>settlingTime</i>	<a href="#">“Settling Time Function” on page 85</a>	<a href="#">“Settling Time Function” on page 134</a>
<i>slewRate</i>	<a href="#">“Slew Rate Function” on page 86</a>	<a href="#">“Slew Rate Function” on page 135</a>
<i>spectralPower</i>	<a href="#">“Spectral Power Function” on page 86</a>	<a href="#">“Spectral Power Function” on page 136</a>
<i>stddev</i>	<a href="#">“Standard Deviation (stddev) Function” on page 87</a>	<a href="#">“Standard Deviation (stddev) Function” on page 137</a>
<i>table</i>	<a href="#">“Table Function” on page 87</a>	<a href="#">“Table Function” on page 137</a>
<i>tangent</i>	<a href="#">“Tangent Function” on page 88</a>	<a href="#">“Tangent Function” on page 138</a>
<i>thd</i>	<a href="#">“Total Harmonic Distortion (thd) Function” on page 89</a>	<a href="#">“Total Harmonic Distortion (thd) Function” on page 139</a>
<i>value</i>	<a href="#">“Value Function” on page 90</a>	<a href="#">“Value Function” on page 141</a>
<i>xmax</i>	<a href="#">“xmax and ymax” on page 74</a>	<a href="#">“xmax and ymax” on page 123</a>
<i>xmin</i>	<a href="#">“xmin and ymin” on page 75</a>	<a href="#">“xmin and ymin” on page 124</a>
<i>xval</i>	<a href="#">“X Value (xval) Function” on page 90</a>	<a href="#">“X Value (xval) Function” on page 141</a>
<i>ymax</i>	<a href="#">“xmax and ymax” on page 74</a>	<a href="#">“xmax and ymax” on page 123</a>

## Waveform Calculator User Guide

### RF Mode

---

Function	For RPN mode, see	For algebraic mode, see
<i>ymin</i>	<u>"xmin and ymin"</u> on page 75	<u>"xmin and ymin"</u> on page 124

---

---

## Using the Calculator Special Functions with SpectreRF Simulation Results

This appendix shows you how to use several of the special functions in the Analog Design Environment Waveform Calculator with RF simulation results. It illustrates how to

- Use the `ipn` and `ipnVRI` Calculator special functions in RPN and Algebraic modes to plot IP3.
- Use the `ipn` and `ipnVRI` special functions with scalar inputs.
- Use the `spectralPower` special function to plot spectral power.

This appendix is designed for users of SpectreRF in the Analog Design Environment. It assumes that users are familiar with SpectreRF and its swept `PSS`, `PAC`, and `PSS` analyses.

If you need more information on SpectreRF than is provided in this appendix, see the *SpectreRF User Guide*.

### Using the `ipn` and `ipnVRI` Calculator Special Functions

This example uses an editable copy of the *ne600p* schematic from a copy of the *rfExamples* library. As you follow the example, you

- Perform a swept `PSS` analysis followed by a `PAC` small-signal analysis to produce data for several IP3 plots.
- Plot IP3 using the Direct Plot form.
- Plot IP3 several ways using the Calculator `ipn` and `ipnVRI` special functions.
- Compare the different results.

## Setting Up the Simulation Environment

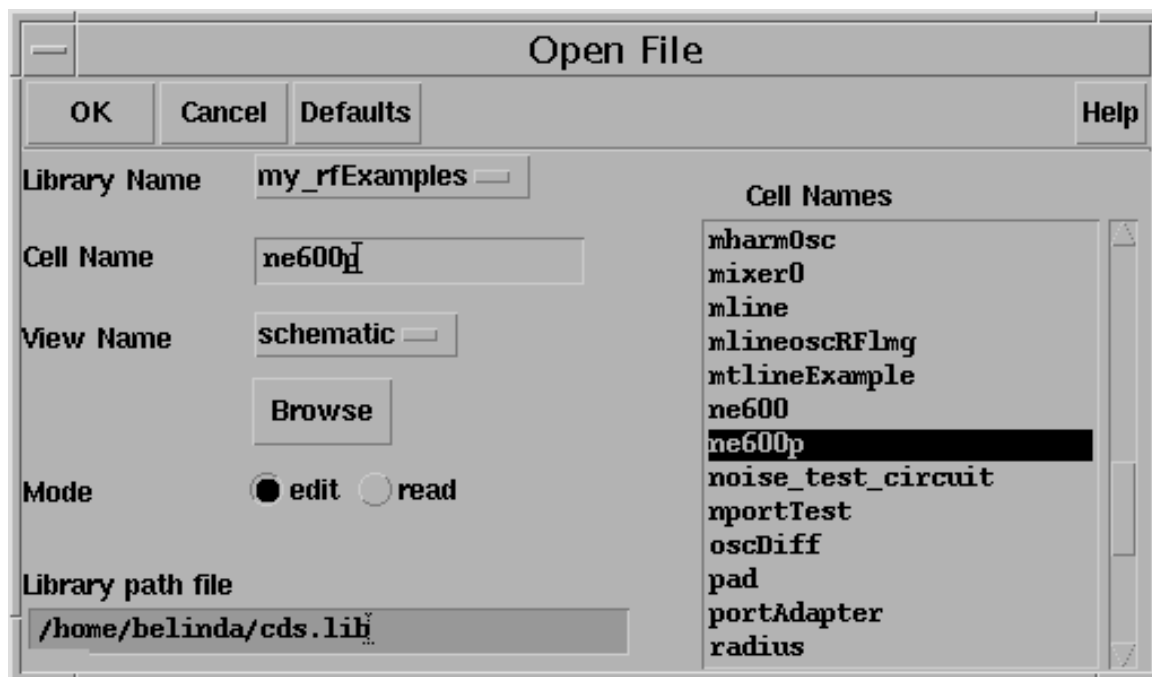
If you need more information than is provided in this appendix, see the *SpectreRF User Guide*. For example, Chapter 3 in the *SpectreRF User Guide* describes in detail how to create the editable copy of the *rfExamples* library of schematics.

### Start the CIW, Open the Schematic, and Set Up the Simulator and Models

1. Start the Command Interpreter Window (CIW) by typing `icms` in a terminal window.

Make sure that you have copied the *ne600p* schematic from the *rfExamples* library in the Cadence installation to your local library of editable schematics (*my\_rfExamples* in this example) so you will have edit permissions on the schematic.

2. In the CIW, choose *File—Open*.
3. In the Open File form
  - a. Select your local library of editable schematics.
  - b. Select the *ne600p* schematic for editing.
  - c. Click *OK*.

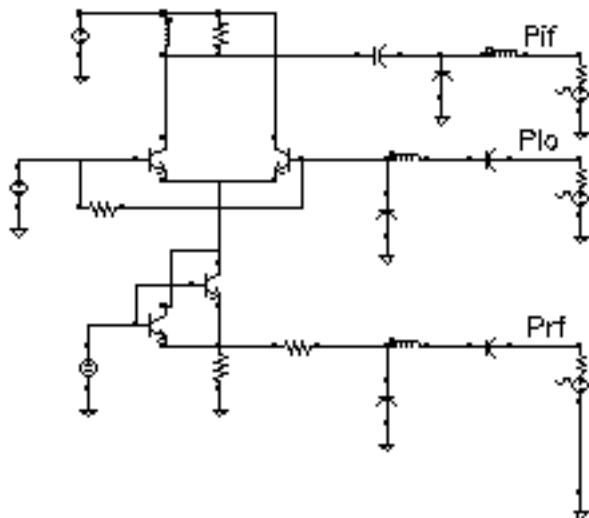


## Waveform Calculator User Guide

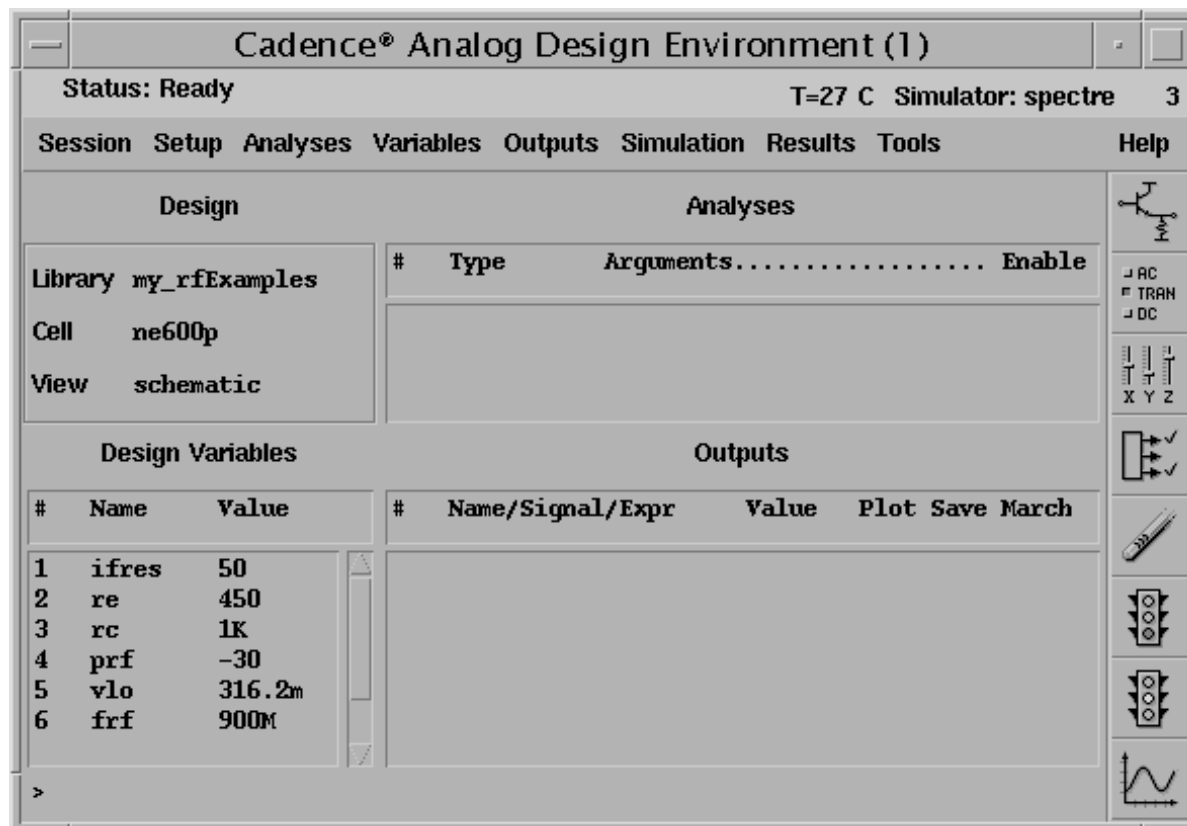
### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The Schematic window opens with the *ne600p* schematic displayed.



4. In the Schematic window, choose *Tools—Analog Environment* to open the Simulation window.



## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

5. In the Simulation window, choose *Setup—Simulator/Directory/Host* to open the Choosing Simulator/Directory/Host form. Make the following selections and click *OK*.

The screenshot shows a dialog box titled "Choosing Simulator/Directory/Host". It has buttons for "OK", "Cancel", "Defaults", and "Help". The "Simulator" field is a dropdown menu set to "spectre". The "Project Directory" field is a text box containing "~/simulation". The "Host Mode" field has three radio buttons: "local" (selected), "remote", and "distributed". The "Host" and "Remote Directory" fields are empty text boxes.

- a. Choose *spectre* in the *Simulator* cyclic field.
  - b. Click *local* to set *Host Mode*.
6. In the Simulation window, choose *Setup—Model libraries* to display the Model Library Setup form.

The screenshot shows a dialog box titled "Model Library Setup". It has buttons for "OK", "Cancel", "Defaults", "Apply", and "Help". The main area is a table with two columns: "Model Library File" and "Section". The first row contains the file path "...7/pink/tools/dfII/samples/artist/models/spectre/rfModels.scs" and an empty section field. Below the table is another section with labels "Model Library File" and "Section (opt.)", each followed by a text box. At the bottom are buttons for "Add", "Delete", "Change", "Edit File", and "Browse...".

7. In the Model Library Setup form

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

- a. In the *Model Library File* field, type the path to the library of models.

<path\_to>/tools/dfII/samples/artist/models/spectre/rfModels.scs

In the example, replace <path\_to> with the installation directory for your Cadence software.

- b. Click *Add*.
- c. Click *OK*.

### Edit the frf Design Variable

1. In the Simulation window, choose *Variables—Edit* to display the Editing Design Variables form and edit the value of the frf variable.

Selected Variable		Table of Design Variables		
Name	Value (Expr)	#	Name	Value
frf	920M	1	prf	-30
		2	vlo	316.2m
		3	frf	920M
		4	flo	1G
		5	plo	-10

2. In the Editing Design Variables form
  - a. In the *Table of Design Variables*, click *frf 900M*.

The variable name and value display in the *Name* and *Value (Expr)* fields.
  - b. Set the variable value in the *Value (Expr)* field to 920M and click *Change*.

The modified value displays in the *Table of Design Variables*, *frf 920M*.
  - c. Click *OK*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

The modified variable value displays in the *Design Variables* area of the Simulation window.

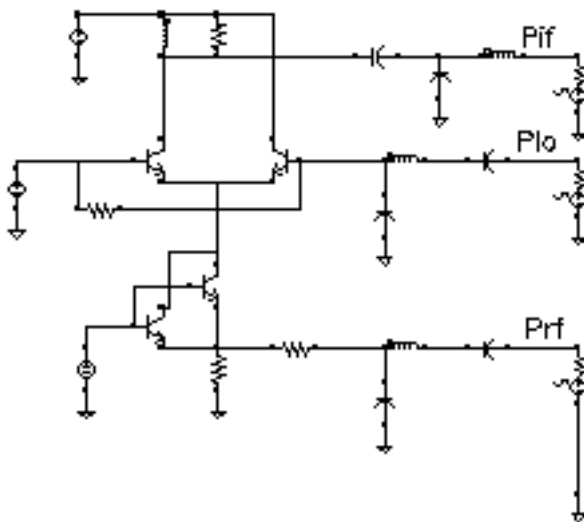
Design Variables		
#	Name	Value
1	ifres	50
2	re	450
3	rc	1K
4	prf	-30
5	vlo	316.2m
6	frf	920M

3. In the Simulation window, if necessary, choose *Analyses—Disable* to disable any analyses you might have run previously.

## Editing the Schematic

Modify and verify the *port* sources in the *ne600p* schematic which is displayed in Figure A-1.

**Figure A-1 The ne600p Schematic**



## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

1. In the Schematic window, click on the *rf port* source to select it.

The *rf port* source is highlighted.

2. Choose *Edit—Properties—Objects* to display the Edit Object Properties form.

The Edit Object Properties form displays with the current information for the *rf port* source.

3. In the Edit Object Properties form for the *rf port*, do the following.

- a. Select *sine* from the *Source type* cyclic field.

- b. Type `frf` in the *Frequency 1* and *Frequency name 1* fields.

- c. Type `prf` in the *Amplitude 1(dBm)* and *PAC magnitude (dBm)* fields. (The *PAC magnitude (dBm)* field is close to the bottom of the form. You may have to scroll down to see it.) This simulation uses dBm rather than magnitude values.

- d. Click *Apply*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The top of the CDF parameter section in the Edit Object Properties form for the *rf port* looks like the following.

CDF Parameter	Value
Resistance	50 Ohms
Port number	1
DC voltage	
Source type	sine <input type="checkbox"/>
Frequency name 1	frf
Frequency 1	frf Hz
Amplitude 1 (Vpk)	
Amplitude 1 (dBm)	prf
Phase for Sinusoid 1	
Sine DC level	
Delay time	
Display second sinusoid	<input type="checkbox"/>
Display modulation params	<input type="checkbox"/>
Display small signal params	<input checked="" type="checkbox"/>
PAC Magnitude	
PAC Magnitude (dBm)	prf

- Click on the *lo port* source to select it and verify its properties.

The Edit Object Properties form changes to display properties for the *lo port* source.

Verify that its *Source type* is set to *sine* and click *Apply*.

5. Click on the *if port* source to select it and verify its properties.

The Edit Object Properties form changes to display properties for the *if port* source.

Verify that its *Source type* is set to `sine` and click *Apply*.

6. Click *OK* in the Edit Object Properties form.

### Check and Save the Design

- In the Schematic window, choose *Design—Check and Save*.

Messages display in the CIW as the schematic is checked and saved.

### Setting Up the Analyses

- In the Simulation window, choose *Analyses—Choose* to display the Choosing Analyses form.

### Setting Up the Swept PSS Analysis

1. In the Choosing Analyses form, click `pss` to display fields for the Periodic Steady State analysis.
2. Below the *Fundamental Tones* list box, if necessary, click *Beat Frequency* and then click the *Auto Calculate* button.

The value `40M` displays in the *Beat Frequency* field.

The `PAC` analysis is responsible for the two tones, and the `PSS` is now a single signal analysis. Consequently, the fundamental frequency is set to `40 MHz`.

3. In the *Output harmonics* cyclic field, select *Number of harmonics* and enter `2` in the *Number of harmonics* field.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

- Click *conservative* for the *Accuracy Defaults (errpreset)* setting. The top of the PSS Choosing Analysis form looks as follows.

**Periodic Steady State Analysis**

**Fundamental Tones**

#	Name	Expr	Value	Signal	SrcId
1	flo	flo	1G	Large	lo
2	frf	frf	920M	Large	rf

☒ Beat Frequency ☐ Beat Period   ☒

**Output harmonics**

**Accuracy Defaults (errpreset)**

☒ conservative ☐ moderate ☐ liberal

**Additional Time for Stabilization (tstab)**

**Save Initial Transient Results (saveinit)** ☐ no ☐ yes

- Click *Sweep*.

The form changes to let you specify data to sweep a variable. You might need to scroll down to display all the sweep information.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

- a. In the *Sweep* cyclic field, select *Variable*.
- b. For *Frequency Variable?*, click *no*.
- c. Click *Select Design Variable*.

The Select Design Variable form displays.

- a. In the Select Design Variable form, select `prf` and click *OK*.

The variable name `prf` displays in the *Variable Name* field on the Choosing Analysis form.

6. Click *Start-Stop* for the *Sweep Range* and then enter `-25` and `5` in the *Start* and *Stop* fields, respectively.

The sweep is skewed downward for a down converter.

7. Click *Linear* for the *Sweep Type*, then click *Step Size* and enter `5` in the *Step Size* field.
8. If necessary, click *Enabled*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The bottom of the PSS Choosing Analysis form looks as follows.

The screenshot shows the bottom portion of a software window titled "PSS Choosing Analysis". It contains several sections for configuring the analysis:

- Oscillator**: A checkbox that is currently unchecked.
- Sweep**: A section with a checked "Sweep" checkbox. It includes a "Frequency Variable?" section with "no" selected (radio button) and "yes" (radio button) unselected. Below this is a "Variable Name" text box containing "prf" and a "Select Design Variable" button.
- Sweep Range**: A section with two radio buttons: "Start-Stop" (selected) and "Center-Span" (unselected). To the right are "Start" and "Stop" text boxes containing "-25" and "5" respectively.
- Sweep Type**: A section with two columns of radio buttons. The first column has "Linear" (selected) and "Logarithmic" (unselected). The second column has "Step Size" (selected) and "Number of Steps" (unselected). To the right of "Number of Steps" is a text box containing "5".
- Add Specific Points**: A checkbox that is currently unchecked.
- Enabled**: A checked checkbox.
- Options...**: A button located at the bottom right of the form.

9. Click *Apply*.

### Setting Up the PAC Small Signal Analysis

1. At the top of the Choosing Analyses form, select `pac` to display fields for the Periodic AC small signal analysis.
2. In the *Frequency Sweep Range (Hz)*, *Freq* field, enter 921M

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

Because the sweep section of the PSS analysis is active, only a single point is supported for the PAC analysis.

3. In the *Sidebands* cyclic field, select *Array of indices*.
4. In the *Additional indices* field enter the values  $-21$  and  $-25$  separated by a space.

Given the following

- ☐ A Fundamental Tone at 40 MHz
- ☐ The LO at 1GHz
- ☐ Two RF tones at 920 MHz and 921 MHz

The sidebands of  $-25$  and  $-21$  represent respectively

- ☐ The first-order harmonic of the IF output at 79 MHz ( $921 - 25 * 40 = 79$ )
- ☐ The third-order harmonic at 81 MHz ( $921 - 21 * 40 = 81$ )

5. If necessary, click *Enabled* for the PAC analysis.

6. The completed PAC section looks as follows.

The screenshot shows a dialog box titled "Periodic AC Analysis". At the top, "PSS Beat Frequency (Hz)" is set to "40M". Below this, there is a section for sweep settings. "Sweeptype" is set to "Absolute" (indicated by a dropdown arrow), and the text "Sweep is Currently Absolute" is displayed. Under "Frequency Sweep Range (Hz)", the "Single-Point" radio button is selected, and the "Freq" is set to "921M". A message states: "Because the sweep section of the PSS analysis is active, only a single point for this analysis is currently supported." Below this is a "Sidebands" section. The "Array of indices" radio button is selected. The "Currently active indices" field is empty. The "Additional indices" field contains "-21 -25". At the bottom left, the "Enabled" checkbox is checked. At the bottom right, there is an "Options..." button.

**Periodic AC Analysis**

PSS Beat Frequency (Hz)

Sweeptype  Sweep is Currently Absolute

Frequency Sweep Range (Hz)

☒ Single-Point ☐ Freq

Because the sweep section of the PSS analysis is active,  
only a single point for this analysis is currently supported.

**Sidebands**

☒ Array of indices ☐

Currently active indices

Additional indices

Enabled ☒

7. Click Apply and then click *OK* in the Choosing Analyses form.

In the Simulation window, the *Analysis* area displays the PSS and PAC information you just entered.

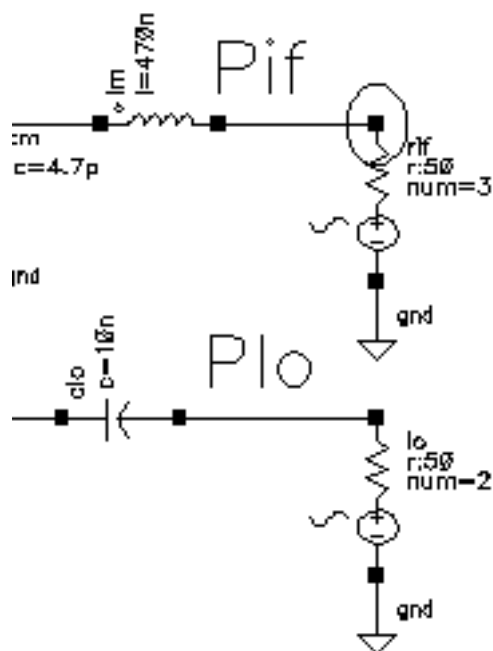
## Selecting Outputs to Save

1. In the Simulation window, choose *Outputs—To Be Saved—Select on Schematic*.

The Schematic window pops to the top.

2. In the Schematic window, click on the positive node of the *rif* port.

The selected node is circled in the schematic. It is also listed in the *Outputs* section of the Simulation window.



## Running the Simulation

In the Simulation window, choose *Simulation—Netlist and Run*.

This example compares two signals only 1 MHz apart.

## Plotting the IP3 Curves Using the Direct Plot Form

1. Select *Results—Direct Plot—Main Form*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The Direct Plot form displays.

**Note:** As you make selections in the Direct Plot form it might change to reflect each selection.

In the Direct Plot form, do the following:

- a. Click on *Replace* for *Plot Mode*.
- b. Click on *pac* for *Analysis*.
- c. Click on *IPN Curves* for *Function*.
- d. In the *Select cyclic* field, select *Port (fixed R(port))*.
- e. Choose *Variable Sweep (prf)* for *Circuit Input Power*.
- f. Type  $-15$  for *Input Power Extrapolation Point (dBm)*.

This value is the intercept point for the ideal amplification extrapolation.

- g. Select the *Input Referred IP3* and *Order 3rd cyclic* fields.
- h. Highlight  $-21\ 81M$  in the *3rd Order Harmonic* list box.
- i. Highlight  $-25\ 79M$  in the *1st Order Harmonic* list box.

**Note:** When you select the *Add to Outputs* button in the Direct Plot form, the *Input Referred IP3* Direct Plot expression is also saved and displayed in the *Outputs* pane in the Simulation Window. You will do this in “Using the IPN Special Function with Scalar Inputs” on page 182.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The completed Direct Plot form looks like the one below.

OKCancelHelp

Plot Mode

☐ Append ☒ Replace

Analysis

☐ pss ☒ pac

Function

☐ Voltage ☐ Current  
☒ IPN Curves

Select

Port ( fixed R(port) )

Circuit Input Power

☐ Single Point  
☒ Variable Sweep ("prf")

"prf" ranges from -25 to 5

Input Power Extrapolation Point (dBm) -15

Input Referred IP3

Order 3rd

3rd Order Harmonic

-25	79M
-21	81M
0	921M

1st Order Harmonic

-25	79M
-21	81M
0	921M

Add To Outputs

☐

> Select Port on schematic...

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

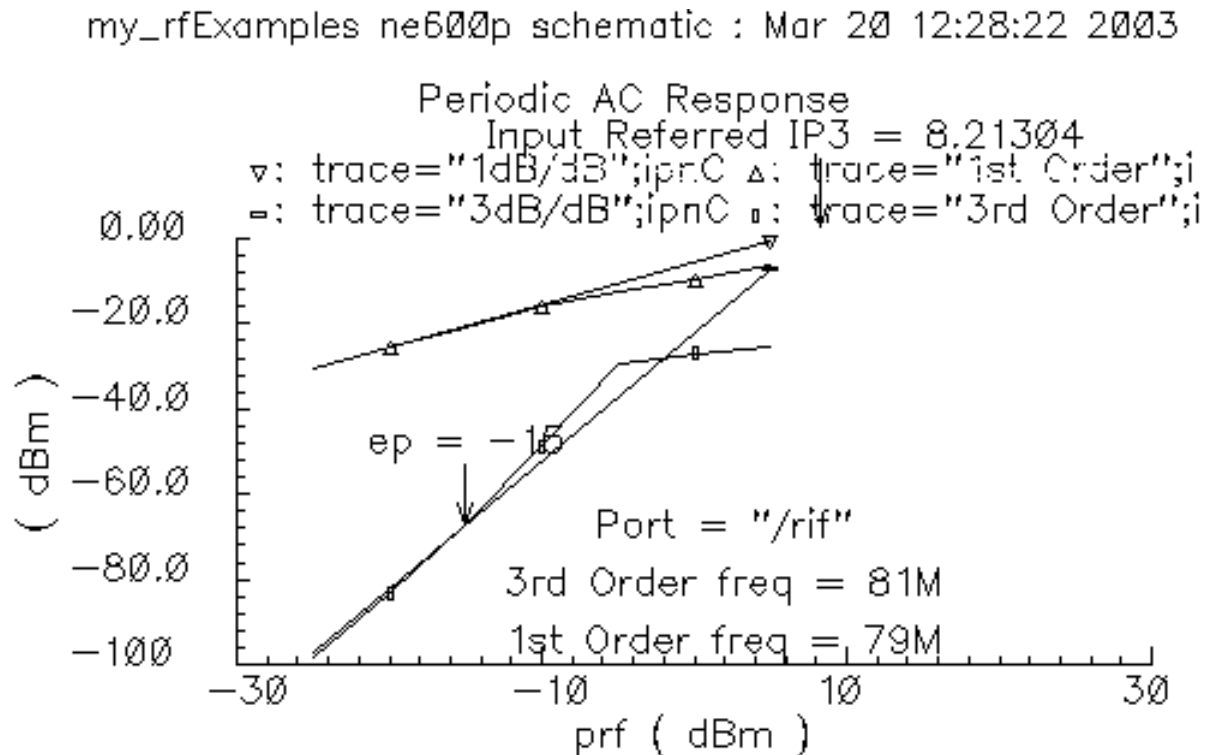
2. Following the prompt at the bottom of the Direct Plot form,

Select Port on schematic...

In the Schematic window, click on the *rif* port.

**Note:** If you click on the *Pif* net (rather than on the *rif* port), you will get an IPN plot with dBV on the vertical axis.

The Waveform window display appears as shown below



**Note:** Notice that the IP3 value and its pointer and label overwrite the legend for the traces in your plot. You can click and drag them above the legend, as in this plot, if you want.

You can also Plot Output Referred IP3 from the same Direct Plot form.

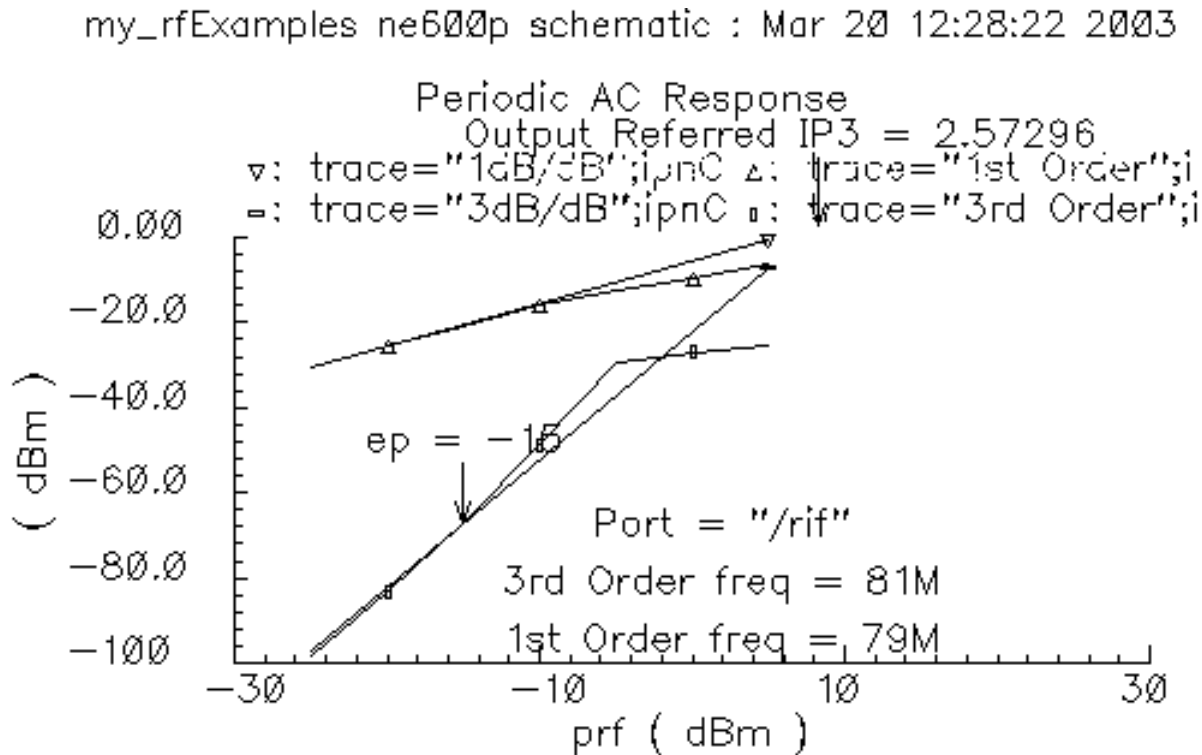
1. Change the cyclic field to *Output Referred IP3* (from *Input Referred IP3*).
2. Click *Replot*.

**Note:** Note that in this case, the measured value is different, depending on whether you select a port (dBm) or a net (dBV) on the schematic.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

Output Referred IP3 for the *rif* port is plotted below.



### Plotting IP3 Using the Calculator *ipn* and *ipnVRI* Special Functions in the RPN Mode

The next two examples use the *ipn* and *ipnVRI* special functions in the Calculator to create IP3 plots. The *ipn* special function calculates only the Input Referred IP3 value. The *ipnVRI* special function calculates both Input Referred IP3 and Output Referred IP3 values.

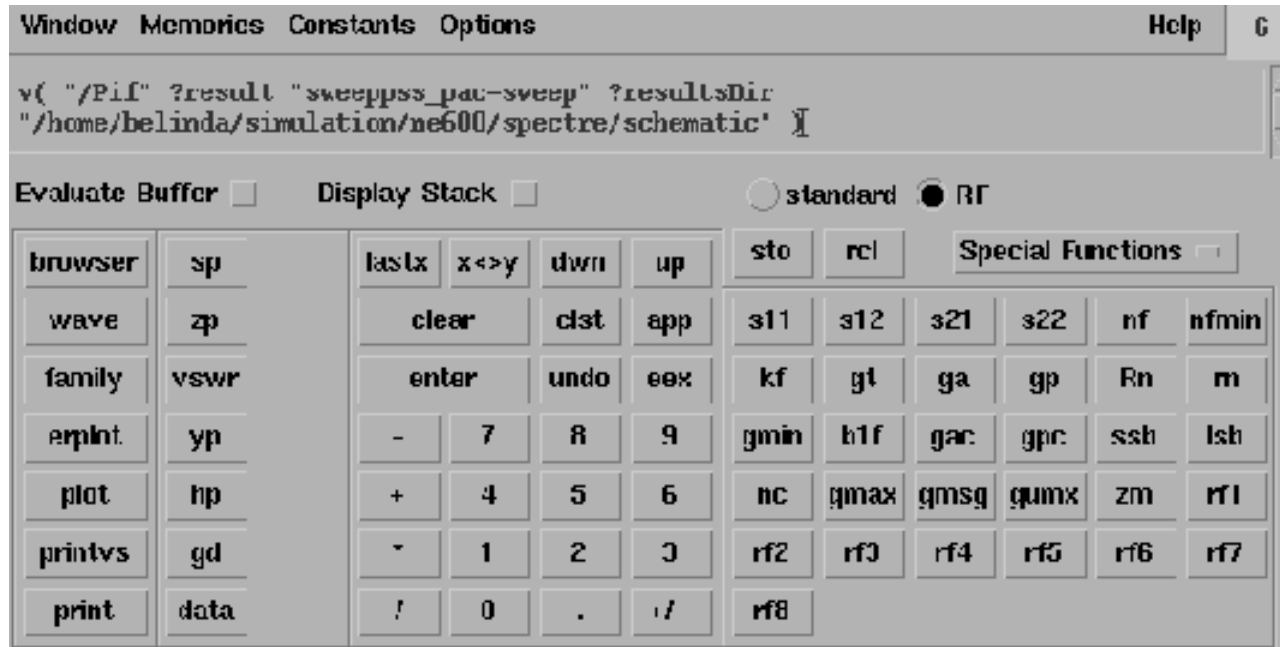
#### Using the *ipn* Special Function in RPN Mode to Plot Input Referred IP3

In this example, you use the *ipn specialFunction* to plot IP3. The *ipn specialFunction* calculates only the *Input Referred IP3*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

1. From the Simulation window, choose *Tools—Calculator*.



2. In this example, you use the Calculator in RPN mode. If the Calculator is in Algebraic mode, change it by selecting *Options—Set RPN*.
3. In the Calculator, click *browser* to open the Results Browser.
4. When the Browse Project Hierarchy form appears, click OK to display the results browser.
5. In the Results Browser, click on:

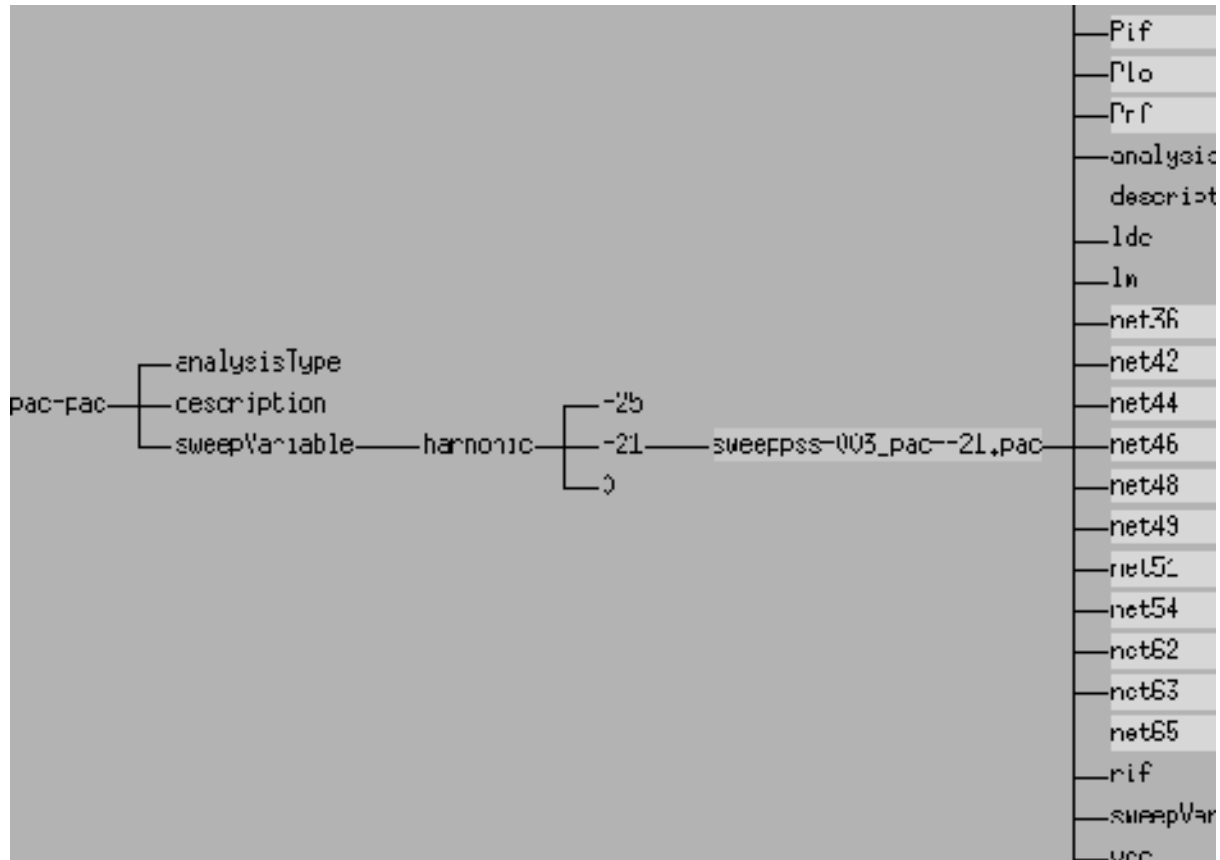
```
schematic->psf->Run1->sweepss_pac-sweep->
sweepVariable->prf-> -10->sweepss-003_pac-pac->
sweepVariable->harmonic-> -21->
sweepss-003_pac--21.pac-> Pif
```

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

Or



The information displays in the Calculator buffer. It looks similar to this:

```
v( "/Pif" ?result "sweepss_pac-sweep" ?resultsDir
"/home/belinda/simulation/ne600p/spectre/schematic" )
```

**Note:** Clicking on any harmonic (–0, –21, or –25) in the results browser gives the same results.

The Results Browser shows the basic structure of the saved simulation. data. Since you choose (in “[Selecting Outputs to Save](#)” on page 167) to save results for the *rif psin* only, the results browser displays only *Pif* data (voltage and current information).

#### 6. In the Calculator, select *Special Functions—ipn*.

The IPN form displays. In the IPN form, do the following:

- a. Enter 3 in the *Spur Order* field.
- b. Enter –21 in the *Spur Harmonic* field.
- c. Enter –15 in the *Extrapolation Point* field.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

- d. Enter -25 in the *Reference Harmonic* field.
- e. Click *OK*.

OK	Cancel	Defaults	Apply	Help
Spur Order	3	Spur Harmonic	-21	
Extrapolation Point	-15	Reference Harmonic	-25	

7. The Calculator buffer contains an expression similar to the bottom lines of the one below:

```
ipn(dB20(harmonic(v( "/Pif" ?result "sweepsspac-sweep" ?resultsDir
"/home/belinda/simulation/ne600p/spectre/schematic" ), -21)),
dB20(harmonic(v( "/Pif" ?result "sweepsspac-sweep"
?resultsDir "/home/belinda/simulation/ne600p/spectre/schematic" ), -25)),
3,1,-15,-15)
```

8. Now you have two choices, either:

- ☐ Click *Evaluate Buffer*

Or,

- ☐ Cut and paste the Calculator buffer into the CIW.

The function evaluates to 8.21304. This number agrees with the IP3 value in the Waveform window.

9. Click *Clear*. if necessary, click *Evaluate Buffer* to deselect it.

The Calculator buffer is cleared.

### Using the ipnVRI Special Function in RPN Mode to Plot IP3

In this example, you use *ipnVRI* on the *specialFunctions* cyclic menu to obtain both Input Referred IP3 and Output Referred IP3 values.

1. Click on the following in the Results Browser:

```
schematic->psf->Run1->sweepsspac-sweep-> sweepVariable->prf->
-10->sweepsspac-003_pac-pac-> sweepVariable->harmonic->
-21->sweepsspac-003_pac--21.pac->Pif
```

The information displays in the Calculator buffer. It looks similar to this:

```
v( "/Pif" ?result "sweepsspac-sweep" ?resultsDir
"/home/belinda/simulation/ne600p/spectre/schematic" )
```

## Waveform Calculator User Guide

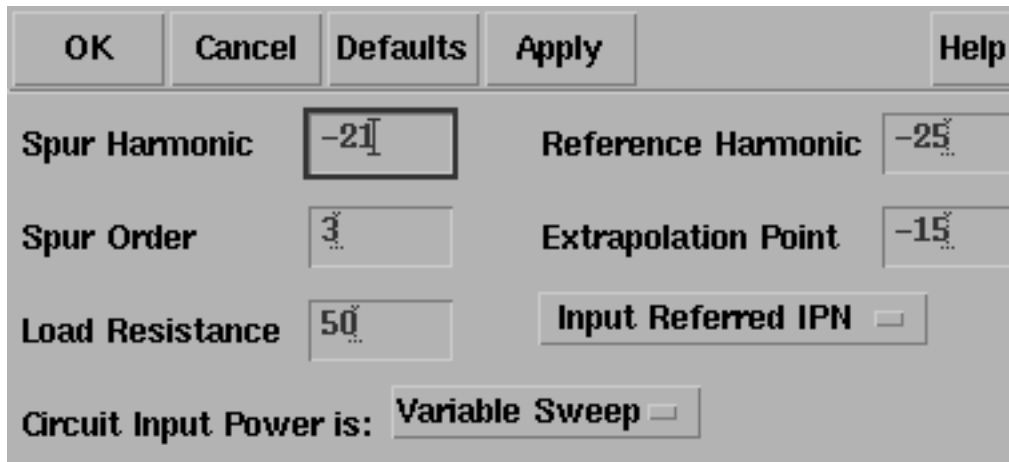
### Using the Calculator Special Functions with SpectreRF Simulation Results

**Note:** You should select a net voltage only when you are using the Calculator. The Load Resistance field is used in conjunction with this voltage to calculate power ( $V^2/R$ ).

**2.** Select *specialFunctions*—*ipnVRI* in the Calculator.

The ipnVRI form displays. In the ipnVRI form, do the following:

- a. Enter -21 in the *Spur Harmonic* field.
- b. Enter -25 in the *Reference Harmonic* field.
- c. Enter 3 in the *Spur Order* field.
- d. Enter -15 in the *Extrapolation Point* field.
- e. Enter 50 in the *Load Resistance* field.
- f. Select *Input Referred IPN* in the cyclic field.
- g. Select *Variable Sweep* in the *Circuit Input Power is:* cyclic field.
- h. Click OK.



**Note:** If you want to plot dBV using the *ipnVRI* function, set the *Load Resistance* to nil.

The Calculator buffer has an expression like the one below:

```
ipnVRI(v( "/Pif" ?result "sweepss_pac-sweep" ?resultsDir  
"/home/belinda/simulation/ne600p/spectre/schematic" ),-21,-25,  
?ordspur 3,?epoint -15,?rport 50)
```

**3.** Click *Evaluate Buffer*.

The expression evaluates to 8.21304166.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

#### Getting Input Referred IP3 Using the Direct Plot Form

There is yet another way to obtain input referred IP3 used by the Direct Plot form:

1. Go to the Waveform Window and select *Annotation—Edit* to display the Annotation form.
2. In the Annotation form list box, click on *Input Referred IP3*. In the expression field, it shows you the expression for that label.

```
ipnVRI(v("/Pif" ?result "pac")-0. '-21 '-25 ?rport  
resultParam("rif:r" ?result "pac") ?epoint -15)
```

3. Click *Get Calculator Expression* to obtain the same IP3 value (8.21304166). The Direct Plot expression uses the *ipnVRI* function, which is used to simplify the declaration of an IPN measurement.

Annotation (window:10)

OK Cancel Help

Choose Label

Text/Expression

1 Input Referred IP3 = ipnVRI((v("/Pif" ?result  
2 ep = -15.000000  
3 Port = sprintf(nil "/rif")  
4 3rd Order freq = cadar(setof(x harmonicFreqlis  
5 1st Order freq = cadar(setof(x harmonicFreqlis

Add Delete Change Clear

Type ☐ Fixed ☐ Curve ☒ Curve (With Arrow) Curve 1 tra...

Contents ☐ text ☒ expression Input Ref = 8.21304166521713

Get Calculator Expression

Orientation ☒ horizontal ☐ vertical Justification lowerCenter

Font Style stick  Size 11 Pen

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

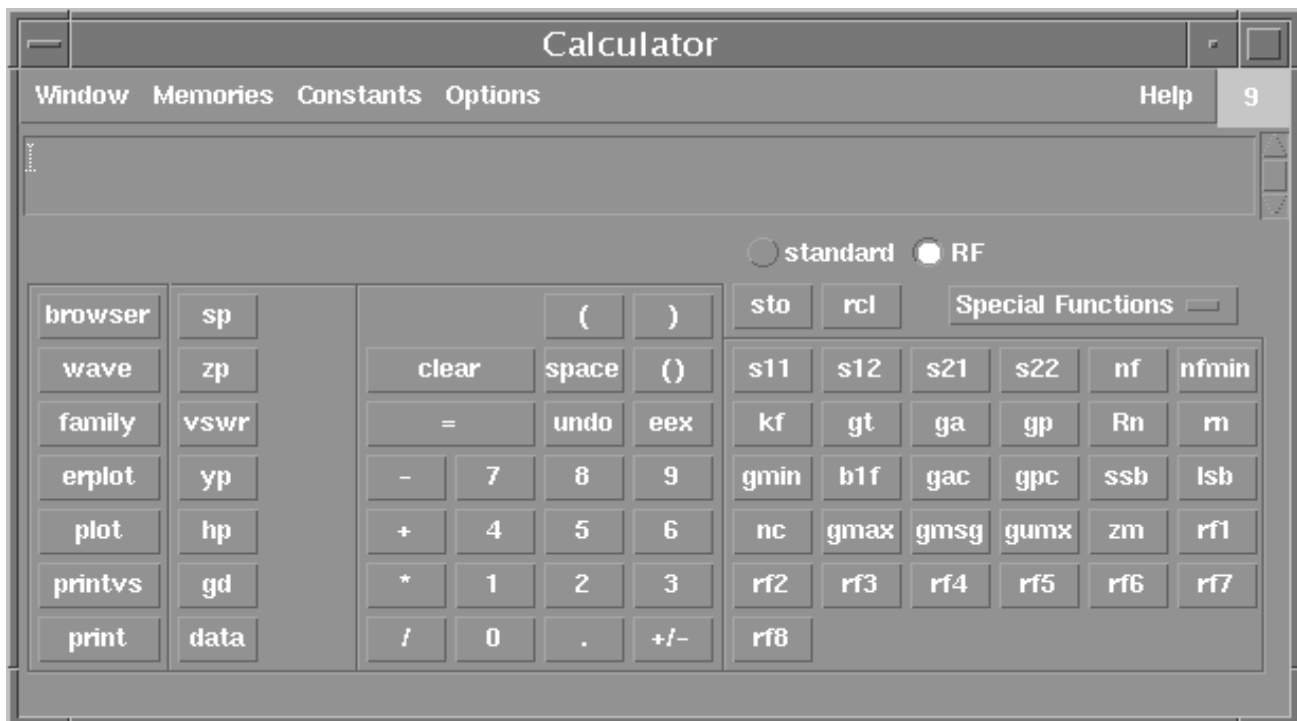
## Plotting IP3 Using the Calculator *ipn* and *ipnVRI* Special Functions in the Algebraic Mode

The next two examples use the *ipn* and *ipnVRI* special functions in the Calculator to create IP3 plots in the Algebraic mode. The *ipn* special function calculates only the Input Referred IP3 value. The *ipnVRI* special function calculates both Input Referred IP3 and Output Referred IP3 values.

### Using the *ipn* Special Function in Algebraic Mode to Plot Input Referred IP3

In this example, you use the *ipn* special function to plot IP3. The *ipn* special function calculates only the *Input Referred IP3*.

1. From the Simulation window, choose *Tools—Calculator*.



2. In this example, you use the Calculator in the Algebraic mode. If the Calculator is in the RPN mode, change it by selecting *Options—Set Algebraic*.
3. In the Calculator, select *Special Functions—ipn*.  
The IPN form displays.
4. In the Calculator, click *browser* to open the Results Browser.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

5. When the Browse Project Hierarchy form appears, click OK to display the results browser.

6. In the Results Browser, click on:

```
schematic->psf->Run1->sweep_pss_pac-sweep->  
sweepVariable->prf-> -10->sweep_pss-003_pac-pac->  
sweepVariable->harmonic-> -21->  
sweep_pss-003_pac--21.pac-> Pif
```

Or



The information displays in the Calculator buffer. It looks similar to this:

```
v( "/Pif" ?result "sweep_pss_pac-sweep" ?resultsDir  
"/hm/uma/simulation/ne600p/spectre/schematic" )
```

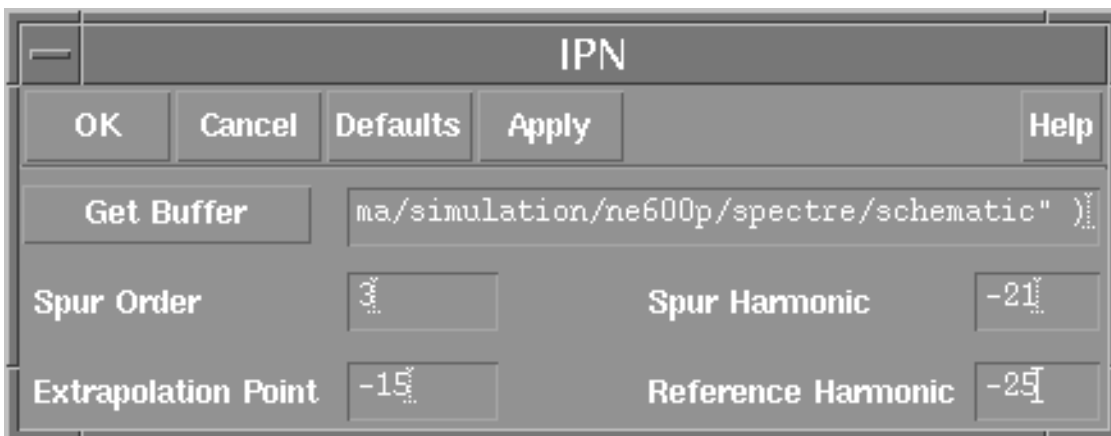
**Note:** Clicking on any harmonic (-0, -21, or -25) in the results browser gives the same results.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

The Results Browser shows the basic structure of the saved simulation. data. Since you choose (in “[Selecting Outputs to Save](#)” on page 167) to save results for the *rif psin* only, the results browser displays only *Pif* data (voltage and current information).

7. In the IPN form, do the following:
  - a. Click *Get Buffer*. The contents of the Calculator buffer appear.
  - b. Enter 3 in the *Spur Order* field.
  - c. Enter -21 in the *Spur Harmonic* field.
  - d. Enter -15 in the *Extrapolation Point* field.
  - e. Enter -25 in the *Reference Harmonic* field.
  - f. Click *Apply* and then click *OK*.



IPN				
OK	Cancel	Defaults	Apply	Help
Get Buffer	ma/simulation/ne600p/spectre/schematic" )			
Spur Order	3	Spur Harmonic	-21	
Extrapolation Point	-15	Reference Harmonic	-25	

8. Click the = button in Calculator to evaluate the expression.

### Using the ipnVRI Special Function in Algebraic Mode to Plot IP3

In this example, you use *ipnVRI* on the *specialFunctions* cyclic menu to obtain both Input Referred IP3 and Output Referred IP3 values.

1. In the Calculator, select *Special Functions—ipnVRI*.

The IPN form displays.
2. In the Calculator, click *browser* to open the Results Browser.
3. When the Browse Project Hierarchy form appears, click OK to display the results browser.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

#### 4. In the Results Browser, click on:

```
schematic->psf->Run1->sweep_pss_pac-sweep->  
sweepVariable->prf-> -10->sweep_pss-003_pac-pac->  
sweepVariable->harmonic-> -21->  
sweep_pss-003_pac--21.pac-> Pif
```

Or



The information displays in the Calculator buffer. It looks similar to this:

```
v( "/Pif" ?result "sweep_pss_pac-sweep" ?resultsDir  
"/hm/uma/simulation/ne600p/spectre/schematic" )
```

**Note:** Clicking on any harmonic (-0, -21, or -25) in the results browser gives the same results.

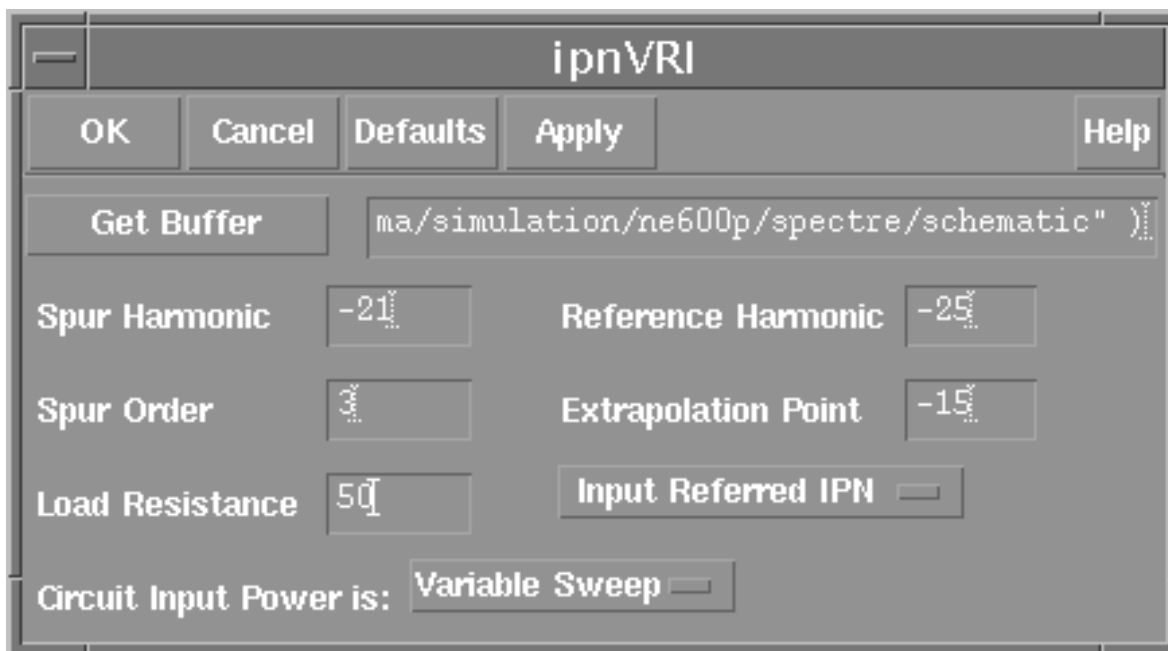
The Results Browser shows the basic structure of the saved simulation. data. Since you choose (in [“Selecting Outputs to Save”](#) on page 167) to save results for the *rif psin* only, the results browser displays only *Pif* data (voltage and current information).

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

5. In the IPN form, do the following:
- Click *Get Buffer*. The contents of the Calculator buffer appear.
  - Enter -21 in the *Spur Harmonic* field.
  - Enter -25 in the *Reference Harmonic* field.
  - Enter 3 in the *Spur Order* field.
  - Enter -15 in the *Extrapolation Point* field.
  - Enter 50 in the *Load Resistance* field.
  - Select *Input Referred IPN* in the cyclic field.
  - Select *Variable Sweep* in the *Circuit Input Power is:* cyclic field.
  - Click *Apply* and then click *OK*.



The screenshot shows the **ipnVRI** dialog box. It features a title bar with the text "ipnVRI". Below the title bar are five buttons: "OK", "Cancel", "Defaults", "Apply", and "Help". Under these buttons is a "Get Buffer" button and a text field containing the path "ma/simulation/ne600p/spectre/schematic" ). The main area of the dialog contains several input fields and dropdown menus. On the left side, there are three rows of labels and input fields: "Spur Harmonic" with the value "-21", "Spur Order" with the value "3", and "Load Resistance" with the value "50". On the right side, there are two rows of labels and input fields: "Reference Harmonic" with the value "-25" and "Extrapolation Point" with the value "-15". Below these is a dropdown menu labeled "Input Referred IPN" which shows a radio button. At the bottom of the dialog, there is a label "Circuit Input Power is:" followed by a dropdown menu showing "Variable Sweep".

**Note:** If you want to plot dBV using the *ipnVRI* function, set the *Load Resistance* to nil.

6. Click the = button in Calculator to evaluate the expression.

## Using the IPN Special Function with Scalar Inputs

This example uses the editable copy of the *ne600p* schematic with scalar, rather than swept, inputs. Two reasons that you might want to use scalar inputs with the IPN function (assuming that the region of operation is linear) are that

- Simulation is faster
- IPN optimization is faster

**Note:** Whenever you use the *ipn* function with scalar inputs, always follow up by doing a final *ipn* verification with a power sweep (non-scalar inputs), as described in [“Using the ipn and ipnVRI Calculator Special Functions”](#) on page 153. This will verify your results.

### Setting Up the Simulation Environment

This example uses the editable copy of the *ne600p* schematic from the *rfExamples* library.

If necessary, start *icms* and edit the *ne600p* schematic as described in the previous example.

- See [“Start the CIW, Open the Schematic, and Set Up the Simulator and Models”](#) on page 154.
- Edit the *frf* design value as described in the previous example. See [“Edit the frf Design Variable”](#) on page 157.
- Edit the *ne600p* schematic. See [“Editing the Schematic”](#) on page 158.
- Finally, see [“Check and Save the Design”](#) on page 161.

### Setting Up the Analyses

Before you set up the analyses, use *Analyses—Disable* in the Simulation window to disable any analyses you may have run previously.

- In the Simulation window, use *Analyses—Choose* to display the Choosing Analyses form.

### Setting Up the PSS Analysis

The PSS analysis is set up similar to the previous example, except that the *Sweep* button is disabled.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

1. In the Choosing Analyses form, click *pss* to display fields for the Periodic Steady State analysis.
2. Below the *Fundamental Tones* list box, if necessary, click *Beat Frequency* and then click *Auto Calculate*.
3. In the *Output harmonics* cyclic field, select *Number of harmonics* and enter 2 in the *Number of harmonics* field.
4. Click on *conservative* for the *Accuracy Defaults (errpreset)* setting.
5. Make sure *Sweep* is not selected.
6. Click *Enabled*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The bottom of the PSS analysis form looks like this

The screenshot shows the bottom section of a PSS analysis form. At the top, there are three buttons: "Clear/Add", "Delete", and "Update From Schematic". Below these are two radio buttons: "Beat Frequency" (selected) and "Beat Period". To the right of the "Beat Frequency" radio button is a text box containing "40M". To the right of the "Beat Period" radio button is a text box containing "Auto Calculate" and a checked checkbox. Below this is a section titled "Output harmonics" with a label "Number of harmonics" and a text box containing "2". Below this is a section titled "Accuracy Defaults (empreset)" with three radio buttons: "conservative" (selected), "moderate", and "liberal". Below this is a label "Additional Time for Stabilization (tstab)" and a text box containing "...". Below this is a label "Save Initial Transient Results (saveinit)" and two radio buttons: "no" and "yes". Below this is a label "Oscillator" and a checked checkbox. Below this is a label "Sweep" and a checked checkbox. At the bottom, there is a label "Enabled" and a checked checkbox, and a button labeled "Options...".

7. Click *Apply*.
8. At the top of the Choosing Analyses form, click *pac*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

9. Set the *Frequency Sweep Range (Hz)* cyclic field to *Single-Point*. The value 921M displays in the *freq* field.
10. In the *Sidebands* cyclic field, select *Array of indices* and type -21 and -25 separated by a space in the *Additional indices* field.

Given the following

- ☐ A Fundamental Tone at 40 MHz
- ☐ The LO at 1GHz
- ☐ Two RF tones at 920 MHz and 921 MHz

The sidebands of -25 and -21 represent respectively

- ☐ The first-order harmonic of the IF output at 79 MHz ( $921 - 25 * 40 = 79$ )
- ☐ The third-order harmonic at 81 MHz ( $921 - 21 * 40 = 81$ )

11. Click *Enabled*.

12. Click *Apply*.

13. The completed PAC section of the Choosing Analyses form looks like this.

The screenshot shows a dialog box titled "Periodic AC Analysis". It contains several input fields and checkboxes. At the top, "PSS Best Frequency (Hz)" is set to "40M". Below this, there is a "Sweep type" dropdown menu with a small square icon next to it, and the text "Sweep is Currently Absolute". Under "Frequency Sweep Range (Hz)", there is a "Single-Point" dropdown menu with a small square icon next to it, and a "Freq" field set to "921M". At the bottom of this section, there is an "Add Specific Points" checkbox. The bottom section of the dialog is titled "Sidebands" and contains an "Array of indices" dropdown menu with a small square icon next to it. Below this, there is a "Currently active indices" field and an "Additional indices" field set to "-21 -25". At the very bottom, there is an "Enabled" checkbox with a small square icon next to it, and an "Options..." button.

14. Click *OK* in the Choosing Analyses form.

## Edit the prf Design Variable

In the Simulation window, choose *Variables—Edit* and change the value of the *prf* design variable to -15.

This value (–15) is the extrapolation value used for the IPN plot in the previous example. See [“Edit the frf Design Variable”](#) on page 157 for information on using the Edit Variables form.

## Running the Simulation

- Select *Simulation—Netlist and Run*.

## Plotting the Single Point IP3 Using the Direct Plot Form

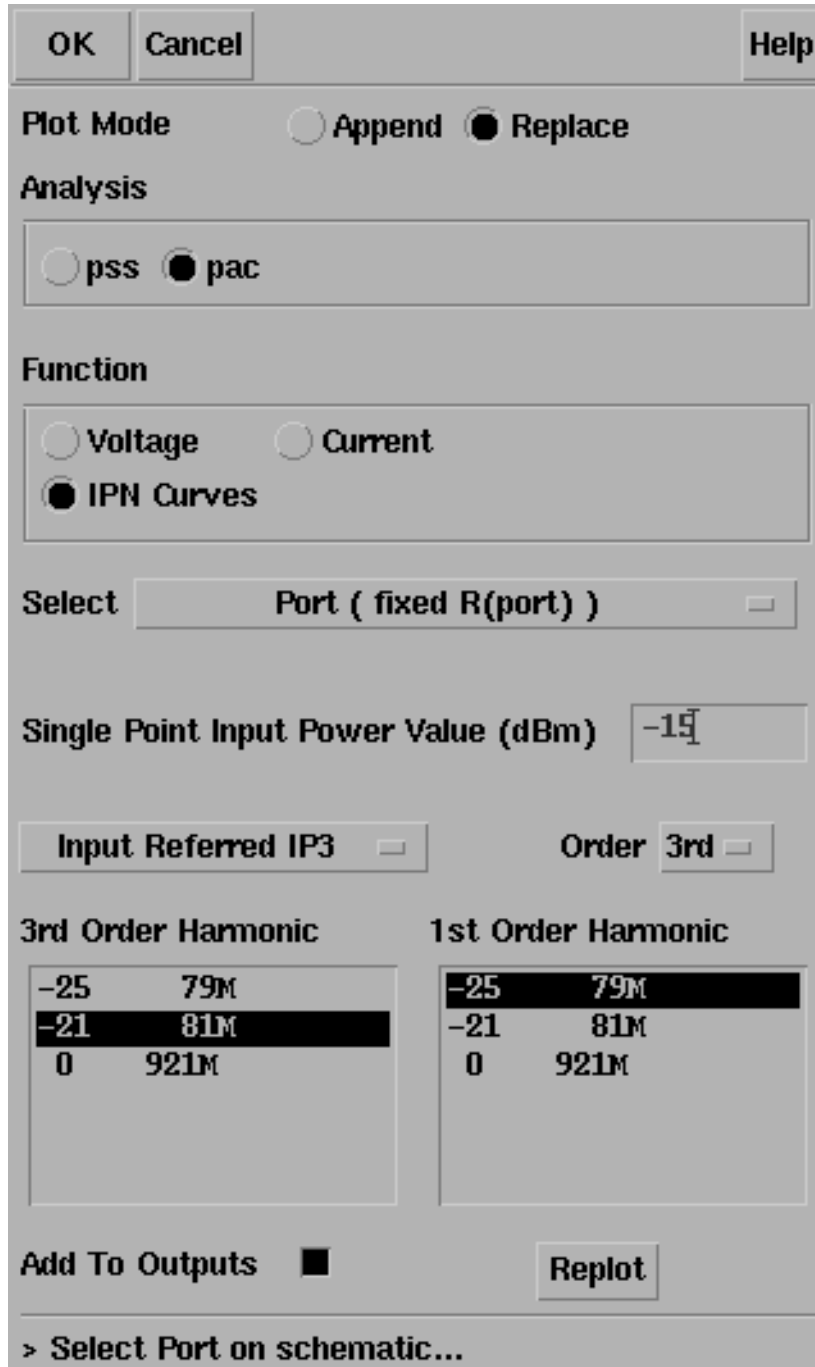
1. After simulation, select *Results—Direct Plot—Main Form*. Enter the values in the PSS Results form as follows:
  - a. Click on *Replace* for *Plot Mode*.
  - b. Click on *pac* for *Analysis Type*.
  - c. Click on *IPN Curves* for *Function*.
  - d. In the *Select cyclic* field, select *Port (fixed R(port))*.
  - e. Type –15 for *Single Point Input Power Value*.
  - f. Select *Input Referred IP3* and *Order 3rd*.
  - g. Highlight –21 81M in the *3rd Order Harmonics* list box.
  - h. Highlight –25 79M in the *1st Order Harmonic* list box.
  - i. Click *Add To Outputs*.
  - j. Following the prompt at the bottom of the Direct Plot form, in the Schematic window, click on the *rif* port

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The Direct Plot form looks like this.



OK Cancel Help

Plot Mode ☐ Append ☒ Replace

Analysis

☐ pss ☒ pac

Function

☐ Voltage ☐ Current

☒ IPN Curves

Select Port ( fixed R(port) )

Single Point Input Power Value (dBm) -15

Input Referred IP3 Order 3rd

3rd Order Harmonic

-25	79M
-21	81M
0	921M

1st Order Harmonic

-25	79M
-21	81M
0	921M

Add To Outputs ☒ Replot

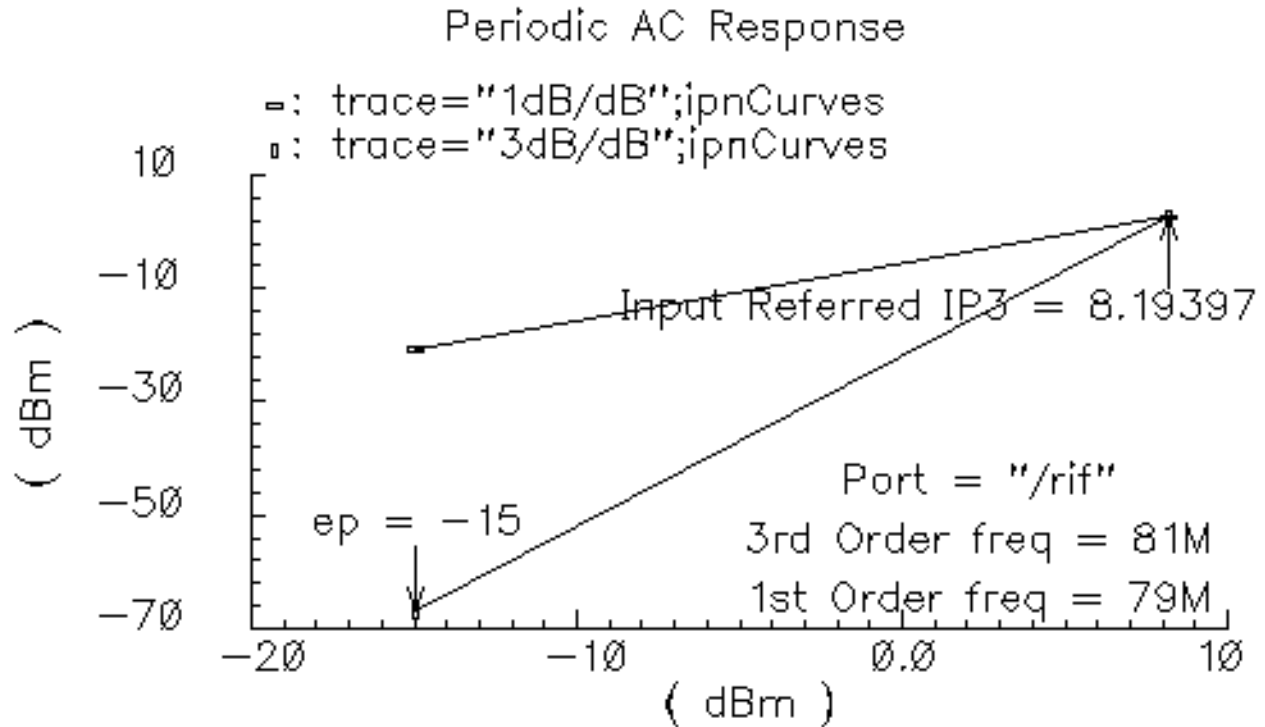
> Select Port on schematic...

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

The single point IP3 is plotted in the waveform window.

my\_rfExamples ne600p schematic : Mar 20 13:56:53 2003



Because you selected the *Add to Outputs* button in the PSS Results form, the Input Referred IP3 Point direct plot expression is also saved to the *Outputs* pane in the Simulation Window

Outputs				
#	Name/Signal/Expr	Value	Plot	Save March
1	rif/PLUS		no	yes no
2	ipnCurves		no	
3	Input Referred IP3..		no	

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

2. If you double click on the *Input Referred IP3...* entry in the Simulation window, the Setting Outputs form appears with *Input Referred IP3...* selected. The Table of Outputs on the right side of the form is reproduced from the Simulation window.

OK Cancel Apply

**Selected Output**

Name (opt.) Input Referred IP3 Point

Expression ("rif:r" ?result 'pac') ?epoint -15

Calculator Open Get Expression Close

Will Be ☐ Plotted/Evaluated

**Table Of Outputs**

#	Name/Signal/Expr	Value
1	rif/PLUS	
2	ipnCurves	
3	Input Referred IP3..	

Add Delete Change Next New Expression

3. Click on *Get Expression* to enter the IP3 into the Calculator buffer or CIW window and obtain the value 8.21304.

```
ipnVRI((v("/Pif" ?result "pac") - 0.0) '-21 '-25 ?rport  
resultParam("rif:r" ?result "pac") ?epoint -15)
```

**Note:** You can copy the expressions stored in the *Outputs* section of the Simulation Window into the *Goals* section of the Optimizer. Use Tools—*Optimizer* in the Simulation window to open the Optimizer. In the Optimizer, choose *Goals—Retrieve Outputs* menu function.

4. The procedure for using the Calculator *ipn* and *ipnVRI* specialFunctions are essentially the same as described in the previous example.

## Using the spectralPower Special Function

This example uses an editable copy of the *ne600p* schematic copied from the *rfExamples* library. As you follow the example, you

- Perform a swept PSS analysis to produce data for several spectral power plots.
- Plot spectral power using the Direct Plot form
- Plot spectral power using the Calculator *spectralPower* special function
- Compare the results

## Setting Up the Simulation Environment

If you need more information than is provided in this appendix, see the *SpectreRF User Guide* for information. For example, Chapter 3 in the *SpectreRF User Guide* describes in detail how to create the editable copy of the *rfExamples* library of schematics.

This example uses the editable copy of the *ne600p* schematic from the *rfExamples* library.

If necessary, start *icms* and edit the *ne600p* schematic as described in the first example in this chapter.

- See “[Start the CIW, Open the Schematic, and Set Up the Simulator and Models](#)” on page 154.
- Edit the *frf* design value as described in the first example. See “[Edit the frf Design Variable](#)” on page 157.
- Edit the *ne600p* schematic. See “[Editing the Schematic](#)” on page 158.
- Finally, see “[Check and Save the Design](#)” on page 161.

## Setting Up the PSS Analysis

Before you set up the PSS analyses, use *Analyses—Disable* in the Simulation window to disable any analyses you may have run previously.

1. In the Simulation window, choose *Analyses—Choose* to display the Choosing Analyses form.
2. In the Choosing Analyses form, select `pss` to display fields for the Periodic Steady State analysis.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

3. Below the *Fundamental Tones* list box, with *Beat Frequency* selected, click the *Auto Calculate* button.

The value 40M is specified as the *Beat Frequency*. Consequently, the fundamental frequency is set to 40 MHz.

4. In the Output harmonics cyclic field, select *Number of harmonics* and enter 2 in the field.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

- Click on `conservative` for the *Accuracy Defaults (errpreset)* setting. The top of the PSS Choosing Analysis form looks as follows.

**Periodic Steady State Analysis**

**Fundamental Tones**

#	Name	Expr	Value	Signal	SrcId
1	flo	flo	1G	Large	lo
2	frf	frf	920M	Large	rf

☒ Beat Frequency   ☒

☐ Beat Period

**Output harmonics**

**Accuracy Defaults (errpreset)**

☒ conservative ☐ moderate ☐ liberal

**Additional Time for Stabilization (tstab)**

**Save Initial Transient Results (saveinit)** ☐ no ☐ yes

- Click on the Sweep button.

The form changes to let you specify data for the variable sweep.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

- a. In the *Sweep* cyclic field, select *Variable*.
- b. For *Frequency Variable?*, click *no*.
- c. Click on the *Select Design Variable* button.
- d. In the Select Design Variable form, click *prf* and click *OK*.

The variable name *prf* displays in the *Variable Name* field on the Choosing Analysis form.

- 7. Click *Start-Stop* for the *Sweep Range* and then enter -25 and 5 for the *Start* and *Stop* values, respectively. The sweep is skewed downward for a down converter.
- 8. Click *Linear* for the *Sweep Type*, then click *Step Size* and enter 5 in the *Step Size* field.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

9. If necessary, click *Enabled*. The bottom of the PSS Choosing Analysis form looks as follows.

The screenshot shows the 'PSS Choosing Analysis' dialog box with the following settings:

- Oscillator**: ☐ (unchecked)
- Sweep**: ☒ (checked)
- Frequency Variable?**: ☒ no, ☐ yes
- Variable**: A dropdown menu showing 'Variable'.
- Variable Name**: A text box containing 'prf'.
- Select Design Variable**: A button.
- Sweep Range**:
  - Start-Stop**: ☒ (selected)
  - Center-Span**: ☐ (unchecked)
  - Start**: A text box containing '-25'.
  - Stop**: A text box containing '5'.
- Sweep Type**:
  - Linear**: ☒ (selected)
  - Logarithmic**: ☐ (unchecked)
  - Step Size**: ☒ (selected)
  - Number of Steps**: ☐ (unchecked)
  - Step Size**: A text box containing '5'.
- Add Specific Points**: ☐ (unchecked)
- Enabled**: ☒ (checked)
- Options...**: A button.

10. Click *Apply*. Then click *OK* in the Choosing Analysis form.

In the Simulation window, the *Analysis* area displays the PSS analysis information you just entered.

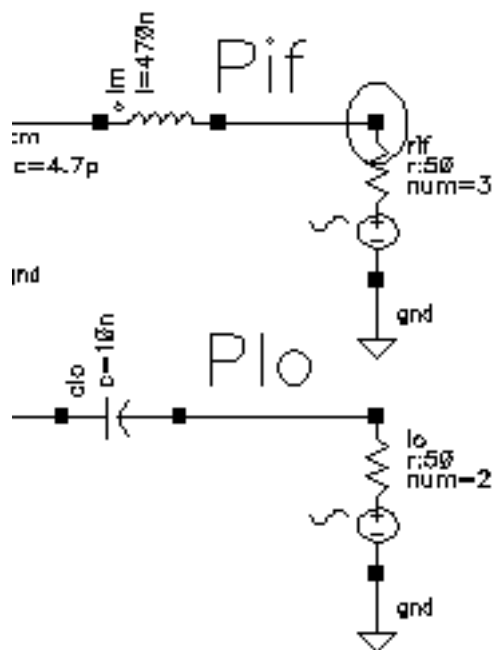
## Selecting Outputs to Save

1. In the Simulation window, choose *Outputs—To Be Saved—Select on Schematic*.

The Schematic window pops to the top.

2. In the Schematic window, click on the positive node of the *rif* psin.

The selected node is circled in the schematic. It is also listed in the *Outputs* section of the Simulation window.



## Running the Simulation

In the Simulation window, choose *Simulation—Netlist and Run*.

## Plotting the Power Spectrum Using the Direct Plot Form

1. Select *Results—Direct Plot—Main Form*.

In the Direct Plot form, do the following:

- a. Click on *Replace* for *Plot Mode*.
- b. Click on *pss* for *Analysis Type*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

- c. Click on *Power* for *Function*.
- d. In the *Select* cyclic field, select *Port (fixed R(port))*.
- e. Click *spectrum* for *Sweep*.
- f. Click dBm for modifier
- g. Highlight *-10* in the *Variable Value (prf)* list box.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The completed form looks like the one below.

The screenshot shows a software interface for the Waveform Calculator. It is configured for a SpectreRF simulation. The 'Plot Mode' is set to 'Replace'. The 'Analysis' is 'pss'. The 'Function' is 'Power'. The 'Select' dropdown is 'Port ( fixed R(port) )'. The 'Sweep' is 'spectrum'. The 'Modifier' is 'dBm'. The 'Variable Value (prf)' list shows values from -25 to 0, with -10 selected. There are buttons for 'Add To Outputs' and 'Replot', and a link '> Select Port on schematic...'.

**Plot Mode** ☐ Append ☒ Replace

**Analysis**

☒ pss

**Function**

☐ Voltage ☐ Current  
☒ Power ☐ Voltage Gain  
☐ Current Gain ☐ Power Gain  
☐ Transconductance ☐ Transimpedance  
☐ Compression Point ☐ IPN Curves  
☐ Power Contours ☐ Reflection Contours  
☐ Harmonic Frequency ☐ Power Added Eff.  
☐ Power Gain Vs Pout ☐ Comp. Vs Pout  
☐ Node Complex Imp.

Select **Port ( fixed R(port) )**

**Sweep**

☒ spectrum ☐ variable

**Modifier**

☐ Magnitude ☐ dB10 ☒ dBm

**Variable Value (prf)**

-25  
-20  
-15  
**-10**  
-5  
0

Add To Outputs ☐ Replot

> Select Port on schematic...

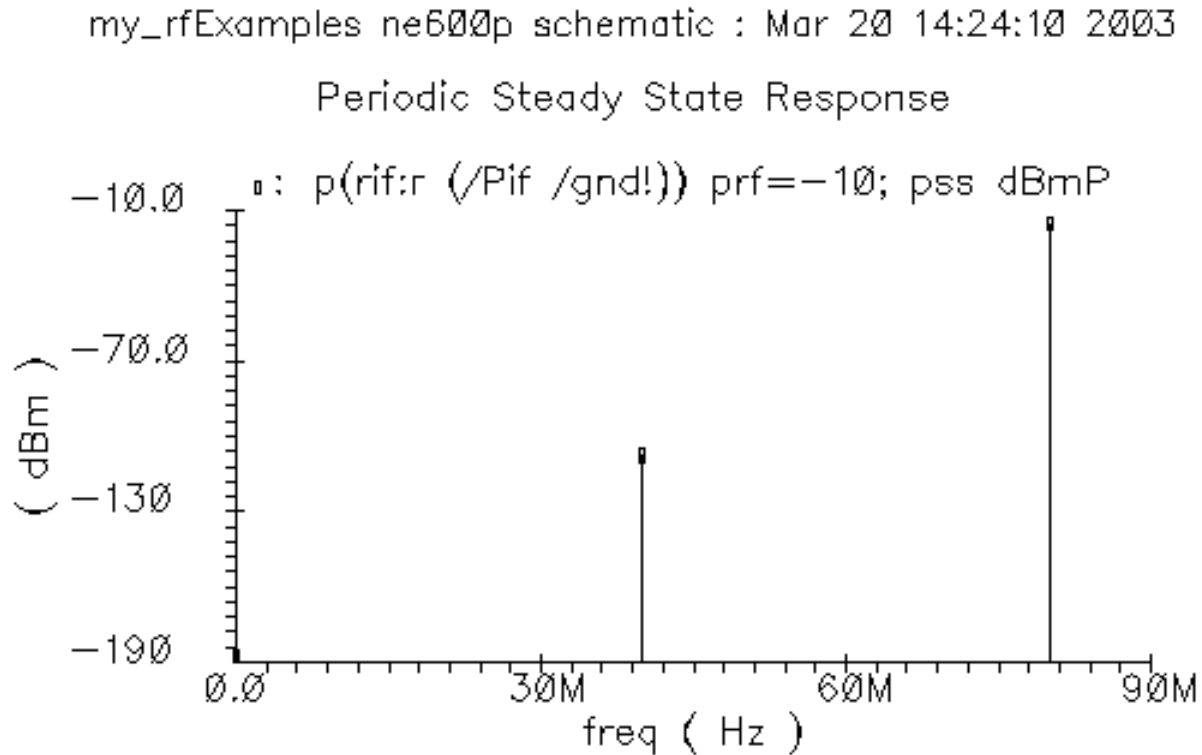
## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

2. Following the prompt at the bottom of the Direct Plot form, in the Schematic window, click on the *rif* port.

The Waveform window display appears as shown below



### Plotting the Power Spectrum Using the Calculator *spectralPower* Special Function

The next example uses the *spectralPower* special function in the Calculator to create spectral power plots.

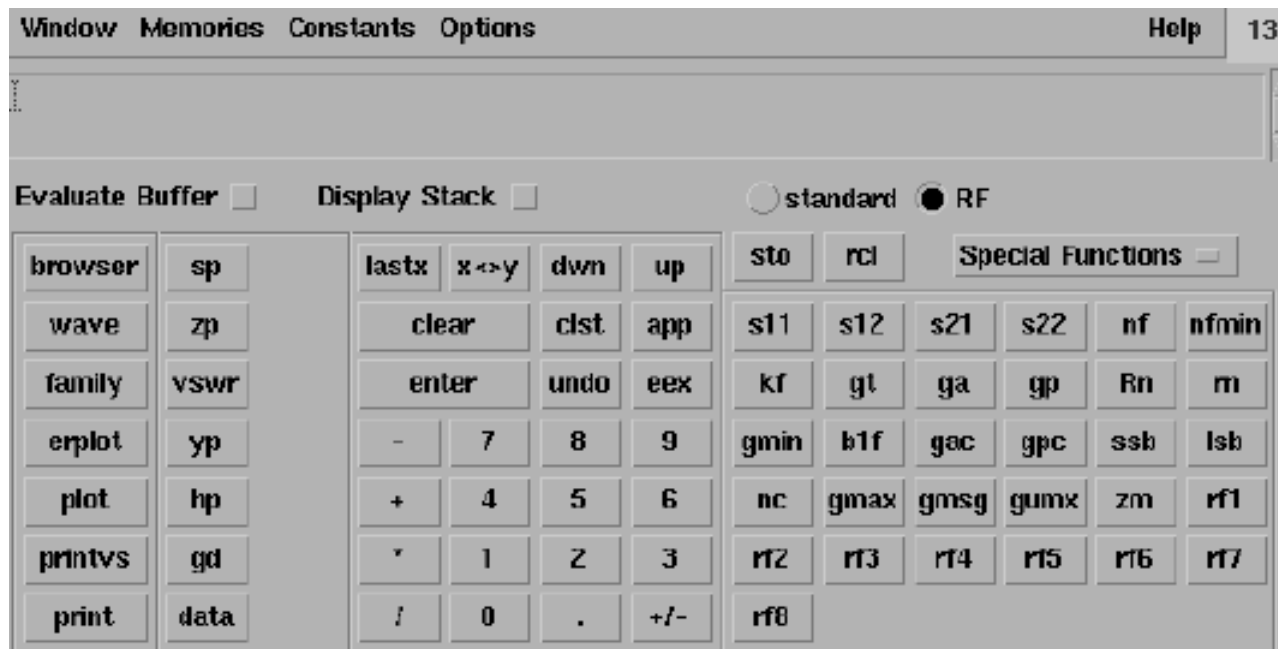
1. From the Simulation window, choose *Tools—Calculator*.

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

---

The Waveform Calculator opens.



2. In this example, you use the Calculator in RPN mode. If the Calculator is in Algebraic mode, change it by selecting *Options—Set RPN*.
3. In the Calculator, click *browser* to open the Results Browser.

When the Browse Project Hierarchy form appears, click *OK* to display the results browser.

## Using the Calculator Special Functions with SpectreRF Simulation Results

```

graph LR
    variable --- prf
    variable --- sweepvss-003_pss-fd.pss
    prf --- -25
    prf --- -20
    prf --- -15
    prf --- -10
    prf --- -5
    prf --- 0
    prf --- 5
    sweepvss-003_pss-fd.pss --- Pif
    sweepvss-003_pss-fd.pss --- Plo
    sweepvss-003_pss-fd.pss --- Prf
    sweepvss-003_pss-fd.pss --- analysis
    sweepvss-003_pss-fd.pss --- descript
    sweepvss-003_pss-fd.pss --- ldc
    sweepvss-003_pss-fd.pss --- lm
    sweepvss-003_pss-fd.pss --- net36
    sweepvss-003_pss-fd.pss --- net42
    sweepvss-003_pss-fd.pss --- net44
    sweepvss-003_pss-fd.pss --- net46
    sweepvss-003_pss-fd.pss --- net48
    sweepvss-003_pss-fd.pss --- net49
    sweepvss-003_pss-fd.pss --- net51
    sweepvss-003_pss-fd.pss --- net54
    sweepvss-003_pss-fd.pss --- net62
    sweepvss-003_pss-fd.pss --- net63
    sweepvss-003_pss-fd.pss --- net65
    sweepvss-003_pss-fd.pss --- q56
    sweepvss-003_pss-fd.pss --- q57
    sweepvss-003_pss-fd.pss --- q58a
    sweepvss-003_pss-fd.pss --- q58b
    sweepvss-003_pss-fd.pss --- rif
    sweepvss-003_pss-fd.pss --- sweepVar
  
```

- ```
schematic->psf->Run1->sweep_pss_pss_fd-sweep->
sweepVariable->prf-> -10->sweep_pss-003_pss-fd.pss-> Pif
```

Product Version 5.0

## Waveform Calculator User Guide

### Using the Calculator Special Functions with SpectreRF Simulation Results

```
v( "/Pif" ?result "sweep_pss_fd-sweep" ?resultsDir
"/ home/belinda/simulation/ne600p/spectre/schematic" )
```

- b. To select the spectral current waveform, in the Results Browser, click on:

```
schematic->psf->Run1->sweep_pss_fd-sweep->
sweepVariable->prf-> -10->sweep_pss-003_pss_fd.pss-> rif
```

The information displays in the Calculator buffer. It looks similar to this:

```
i( "/rif/PLUS" ?result "sweep_pss_fd-sweep" ?resultsDir
"/home/belinda/simulation/ne600p/spectre/schematic" )
```

The voltage waveform previously displayed in the Calculator buffer is passed to the Calculator stack.

5. In the Calculator, select *Special Functions—spectralPower*.
6. In the Calculator, select *Special Functions—dBm*.
7. In the Calculator, click *plot*.

The power spectrum displays in the Waveform window.

my\_rfExamples ne600p schematic : Mar 20 14:24:10 2003

#### Periodic Steady State Response

