“The What-If Solution for Analog & RF Circuit Design”™

Avista Spectre/XL Demonstration User’s Guide

Avista Design Systems

Version 1B for Microsoft Excel
November 1996

AVISTA
DESIGN • SYSTEMS

Avista Design Systems
710 Figureroa Street
Folsom, California 95630

800-985-6080 Sales
916-985-6080 FAX
916-985-6080 Technical Support

Contents

Demonstration Quick-Start Instructions 1

How to Use Avista Spectre/XL Right Away....1
What You Need to Run Avista Spectre/XL....1
Quick Installation and Operation....1
Trying the Demonstration Circuits....2
Analysis....3
What to Do if Something Does Not Work....3

Overview 5

Introduction....5
What is Avista Spectre/XL?....5
Who Should Use Avista Spectre/XL?....6
What's New in This Release....6
Technical Innovation....6
Components....8
Nonlinear Devices....8
SPICE Compatibility....8
Electrical Results....9
Quiescent Nonlinear (DC) Results....9
Linear Frequency Domain (AC) Results....9
Nonlinear Time Domain (HB) Results....9
Nonlinear Frequency-Translation (HAC) Results....9
Linear Noise Results....9
Frequency-Converted Noise (HNOISE) Results....9
S-Parameter (SP) Results....9
Suitable Nonlinear Circuits....10

Demonstration Circuits 11

It's a Candy-Mint, It's a Breath-Mint, It's Spectre/XL!....11
What's All This Embedding Stuff, Anyhow?....12
Activation Changes Excel Window....12
A Sample Circuit....12
Viewing the Sample Circuit....13
Evaluating the Sample Circuit....13
A Linear Amplifier Circuit....14
Other Circuits....15

Using Avista Spectre/XL 19

Comparing Spectre/XL to SPICE....19
Sweeping....19
Charting and Plotting....19
Frequency Domain....19
Time Domain....19
Noise....20

S-Parameter and Two-Port Analysis....21
When Should I Use SPICE? 21
Using Existing SPICE Netlists....21
Worksheet Functions....21
AC for Small-Signal Response 22
DC for DC Bias-Point....23
HAC for Heterodyned AC (Frequency-Converted Small-Signal Response)....23
HB for Time Domain Response....24
HNOISE for Heterodyned Noise (Frequency-Converted Noise)....25
NOISE for Linear Noise 26
SP for *S*-Parameter and Two-Port Analysis 27
LABEL for Identifying Controls....29
Working with Arrays....30

Order Form **31**

CHAPTER 1

Demonstration Quick-Start Instructions

How to Use Avista Spectre/XL Right Away

The best way to familiarize yourself with *Avista Spectre/XL* is by reading the through the *Avista Spectre/XL Demonstration User's Guide* and exercising each of the example circuits found in the EXAMPLES\SPXL subdirectory of the *Microsoft Excel* home directory. This section covers only a few of the most commonly used capabilities of *Avista Spectre/XL*; please read the rest of the guide to learn about the others.

What You Need to Run Avista Spectre/XL

- Any 80486 or higher PC with at least 8 megabytes of memory and 3 megabytes of disk space.
- One of *Microsoft Excel for Windows 95* version 7.0, *Microsoft Excel for Windows* version 5.0.

Quick Installation and Operation

► **To install Avista Spectre/XL for Microsoft Excel 7.0 for Windows 95**

1. Insert the end of the authoring plug marked COMPUTER into a printer port on your computer.
2. Start *Microsoft Windows* if it is not running.
3. Insert the disk labeled

    Disk 1 - Setup
    Avista Spectre/XL version 1B7 for Microsoft Excel 7.0

    in drive A (or B).
4. Click the **Start** button and then click **Run**.
5. Type **a:setup** (or **b:\setup**).
6. Press ENTER.
7. Follow the Setup instructions on the screen.

► **To install Avista Spectre/XL for Microsoft Excel 5.0**

1. Insert the end of the authoring plug marked COMPUTER into a printer port on your computer.
2. Start *Microsoft Windows* if it is not running.

3. Insert the disk labeled

   Disk 1 - Setup
   Avista Spectre/XL version 1B5 for Microsoft Excel 5.0

   in drive A (or B).
4. Choose **Run** from the **File** menu in Program Manager.
5. Type **a:setup** (or **b:\setup**).
6. Press ENTER.
7. Follow the Setup instructions on the screen.

► **To start Microsoft Excel and Avista Spectre/XL**

1. Double-click the *Microsoft Excel* program icon.

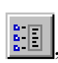
Trying the Demonstration Circuits

Points to Remember The following points need to be remembered:

- Identifiers and parameters in circuit descriptions are case-sensitive.
- *Avista Spectre/XL* links cells on the worksheet with parameters in the circuit description. This link is called a *control*. In the demonstration version these controls have been put in the example circuits for you. You cannot add or delete a control in the demonstration version.
- Worksheet functions return the DC, small-signal and nonlinear results. You can add or delete *Avista Spectre/XL* worksheet functions in the demonstration version.
- Double-click on an *Avista Spectre/XL* circuit object to view the circuit description (netlist). The demonstration version does not allow changes to the netlist or the creation of a new circuits.
- The circuit editor (just a viewer in the demonstration version) remains active until you click outside the circuit object or press the ESC key.

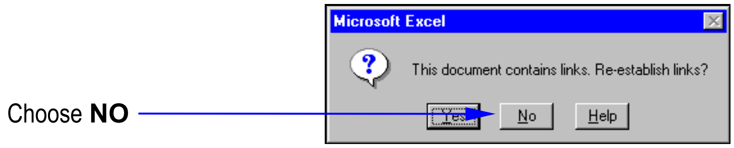
Note Always choose **NO** when *Excel* asks to re-establish links when you open the example workbooks. If you save the workbook after opening it, this request will not appear again.

► **To open an Spectre/XL demonstration workbook**

1. Start *Excel*. The *Excel* window appears. If necessary, enlarge the window by clicking the Maximize button in the upper-right corner of the window.
2. Choose **Open** from the **File** menu.
3. Navigate to the EXAMPLES\SPXL directory which is a subdirectory of where *Excel* is installed (usually \MSOFFICE\EXCEL).
4. Excel 7.0: Click ,the Properties button, to view the summary of a demo file before opening it. Click on each of the demo files to see the summary comments for that workbook. In the **Listed Files** box, select one of the demos listed, for example, RFdemo_1.XLS.

   EXCEL 5.0: Click **Find File** to view a demo file before opening it. Set **View** entry (bottom left corner of the dialog box) to **Summary**. Click on each of the demo files to see the summary comments for that workbook. In the **Listed Files** box, select one of the demos listed, for example, RFdemo_1.XLS.
5. Click the **Open** button.

6. *Excel* will ask whether to re-establish links as shown. Choose **NO**.



7. Wait several seconds for the worksheets in the workbook to be recalculated. Click on the tab labeled "About Demo" for more explanation. The cells outlined in blue change the circuit.

Analysis

The example workbooks provided with the demonstration package conveniently illustrate the analysis features of *Avista Spectre/XL*. You can add your own analyses to these sheets. For more information on adding your own analyses see Chapter 4 "Using Avista Spectre/XL."

What to Do if Something Does Not Work

Call Avista Design Systems Technical Support at 916-985-6080. Please send a fax if you wish to contact Technical Support outside of the hours of 9:00 AM to 5:00 PM Pacific Time.

This page is intentionally left blank.

C H A P T E R 2

Overview

Introduction

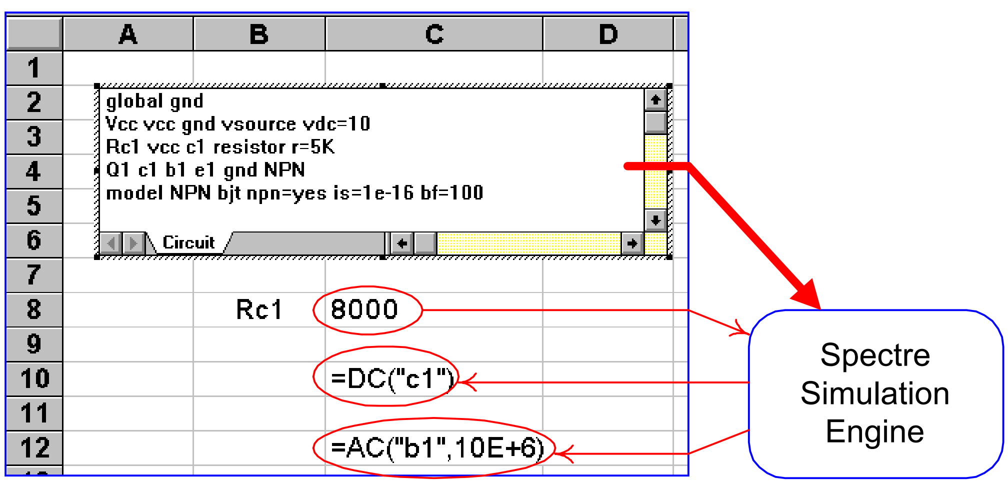
What is Avista Spectre/XL?

Avista Spectre/XL brings accurate circuit analysis to *Microsoft Excel*, the most popular spreadsheet for *Microsoft Windows*.

Powerful Integration

Avista Spectre/XL enhances *Microsoft Excel*'s interactive what-if analysis by integrating the *Spectre*† simulation engine. From within *Microsoft Excel*, you embed a circuit "object", with a SPICE-like description, onto a worksheet (spreadsheet). You then set-up links between worksheet cells and component values in the circuit. Changing the cell value, by hand, by formula, or by programming‡, automatically changes the circuit value. Worksheet functions return nonlinear DC (operating point), linear AC (small signal), nonlinear time domain§ (harmonic), linear noise and *s*-parameter responses. Like any other *Microsoft Excel* worksheet function, you can use *Avista Spectre/XL* worksheet functions in calculations, organized into tables or charted.

Avista Spectre/XL is a responsive and accurate what-if tool for analog circuit design.



Spectre/XL Works Inside Microsoft Excel

† *Spectre*, Version 1a1, Dept. of Electrical Engineering and Computer Sciences, University of California, Berkeley.

‡ Programming can be *Microsoft Excel* macro scripts or Visual Basic for Applications (VBA) code.

§ Nonlinear time domain response is used for the accurate evaluation of long latency and high-frequency circuits that are tough to analyze with SPICE-like time domain simulators.

Who Should Use Avista Spectre/XL?

The Novice

Easy to Use

Avista Spectre/XL is easy to learn. With only modest *Microsoft Excel* skills, you can analyze analog circuits without learning a separate, obscure, CAD program. Many computer users have already worked with spreadsheet programs so they will find *Avista Spectre/XL* easy to learn. If you have no previous spreadsheet experience, time spent learning *Microsoft Excel* may be valuable since the fields of engineering and science often use spreadsheets for number crunching.

The Expert

Tough to Beat

Avista Spectre/XL lets you build better circuits faster. From within the flexible spreadsheet environment, multiple circuit ideas and system calculations can be explored and compared. You can easily save, export, and reuse your completed circuits, or share them with other designers. *Avista Spectre/XL*'s integration into *Microsoft Office* makes documentation easier, and more accurate since analysis data, graphs, and circuit descriptions are only a drag-and-drop away from a report or presentation.

What's New in This Release

- Fast simulation of oscillator circuits.
- MOSFET and advanced bipolar device support.
- Linear Noise analysis.
- *S*-parameter analysis and support for Smith chart circles.
- Circuit temperature control.
- Support of both 16-bit and 32-bit versions of *Microsoft Excel*.

If you purchase *Spectre/XL for RF & Microwave Design*, then the following additional features are available:

- Fast analysis of the intermodulation products and converted noise of mixers, multipliers, and receivers.
- Library of over 100 nonlinear devices from manufacturers such as Harris, Hewlett-Packard, Motorola and NEC

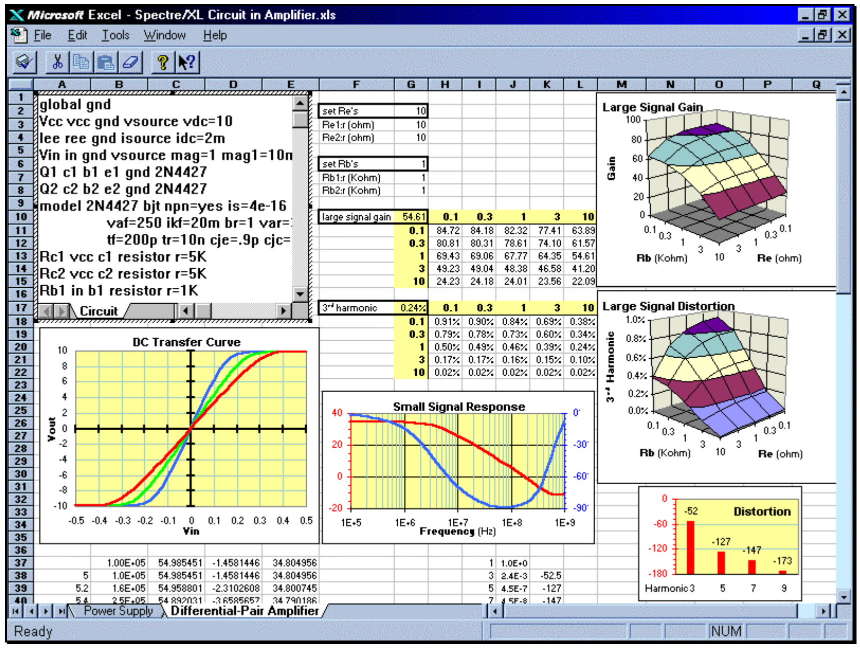
Technical Innovation

Avista Spectre/XL provides a 'home base' for both circuit simulation and design analysis. *Avista Spectre/XL* extends the workbench metaphor of traditional simulator, like SPICE, by providing open access to the simulation engine. Traditional simulators lock you into fixed formats for how you use them. Instead of fighting the simulator to get answers, *Avista Spectre/XL* can customize analysis and presentation for the problem at hand.

Live Analysis

Avista Spectre/XL shortens design time through efficient "live" analysis, which eliminates the *edit-simulate-postprocess-view* cycle of "seamlessly integrated" tools. Live interaction with a circuit idea lets you improve your design quickly, and invites you to explore more design alternatives.

In *Avista Spectre/XL,* you combine any circuit response with *Microsoft Excel* formulas to provide high-level answers about the performance of your design. For example, it would be difficult to imagine using SPICE to generate the results and chart 3rd harmonic distortion versus signal gain, for a bipolar differential pair, as a function of emitter resistance. This is easy to do in *Avista Spectre/XL*.



Avista Spectre/XL encapsulates circuits for design improvement and re-use. A single *Microsoft Excel* file captures your design for easy distribution to other designers, who may develop later generations of the design. Since *Avista Spectre/XL* allows flexible selection of design goals and can make performance enhancements for you, a single circuit design may be quickly re-targeted, or tuned, to new design requirements.

What-If Analysis

What-If Methods

Avista Spectre/XL takes advantage of *Microsoft Excel*'s what-if analysis methods that provide greater insight and easy improvement of your designs.

- **Data Tables** See the effects of varying one or more components in your circuit. For example, a table of response versus frequency versus component value. Create 3-D charts of your table data.
- **Goal Seek Command** Answers simple what-if questions, such as, "What value does R2 need to be to produce the required voltage at this node?" The Goal Seek command solves linear problems.
- **Solver** Answers what-if questions, such as, "What values do R5 and C4 need to be to maximize gain, and keep Re greater than 50." Solver tackles complex, nonlinear problems.
- **Scenario Manager** Save and review design alternatives, and keep track of your different trials.

Sweeping

Policy-Free Sweeps

In SPICE, sweeping is a part of the .DC statement. In many commercial versions, a .STEP statement is also included for varying a component across multiple small-signal (`.AC`) and transient (`.TRAN`) runs as well. Since SPICE combines a sweep and circuit measurement into one operation or statement, it restricts the kind and number of sweeps available.

Spectre/XL separates the sweep feature from the linear and non-linear evaluation of the circuit. You can vary any number and all types of parameters from the worksheet to investigate performance tradeoffs. For example, it is very simple in *Spectre/XL* to sweep two resistors, independently, and chart AC response versus a range of values. This would be difficult to impossible in most versions of SPICE without creating post-processing routines.

Spectre/XL is "policy-free" when it comes to getting the circuit information from your circuit.

Charting and Plotting

Greater insight

You can easily visualize *Avista Spectre/XL* function results with *Microsoft Excel* charts. Not only can circuit performance be plotted across frequency, component changes, and operating conditions, but *Avista Spectre/XL* can easily display 3-D plots of Data Tables.

Using evaluation results in formulas on the worksheet, enables a chart to show not only electrical quantities such as voltage or power, but circuit specifications such as gain, and bandwidth.

Avista Spectre/XL provides a rich selection of over 100 different chart types.

Components

Compatibility

Avista Spectre/XL evaluates circuits containing resistors, capacitors, inductors, transformers, lossless and lossy transmission lines, microstrip lines, independent voltage and independent current sources, voltage-controlled voltage and voltage-controlled current sources, diodes, BJTs, GaAs MESFETs, JFETs, MOSFETs, ports, and N-ports described by *s*-parameter data.

Nonlinear Devices

Avista Spectre/XL uses the same device equations as SPICE for modeling the diode, bipolar junction transistor (BJT), gallium-arsenide field-effect transistor (GaAs MESFET), junction field-effect transistor (JFET), and MOS field-effect transistor (MOSFET). As with SPICE, the models are built-in, so you need only the relevant parameters to describe a device, and you may find these in any SPICE model library.

Device Equations

- The diode model may be used for either junction diodes or Schottky barrier diodes and includes reverse breakdown (Zener) effects.
- The model for the BJT is based on the integral charge model of Gummel and Poon. The model reduces to the simpler Ebers-Moll model if you do not specify the Gummel-Poon parameters. You can include charge storage, parasitic resistances, and current-dependent output conductance with either model.
- The GaAs MESFET model was developed by Statz, Newman, Smith, Pucel and Haus and is completely symmetric.
- The JFET model is based on the FET model of Schichman and Hodges.
- The MOSFET model is the SPICE MOS level 3, the “semi-empirical model”, that includes small-geometry effects.

SPICE Compatibility

Avista Spectre/XL has built-in a translator for importing SPICE netlists into *Avista Spectre/XL* circuit descriptions. It supports the translation of both SPICE2 and SPICE3 netlists. *Avista Spectre/XL* preserves the time and effort you put in developing circuits using SPICE.

The netlist translator is not available in the demonstration version.

Electrical Results

Avista Spectre/XL can calculate the operating point, small-signal, nonlinear frequency, noise and *s*-parameter response of circuits. Furthermore, *Avista Spectre/XL* can perform these tasks while changing supplies, frequency, or any of a large number of device parameters.

Quiescent Nonlinear (DC) Results

Avista Spectre/XL determines the quiescent operation of a circuit by performing a SPICE-type DC bias-point calculation. This static analysis includes the nonlinear behavior of devices and evaluates the state of the circuit with only DC sources applied.

Linear Frequency Domain (AC) Results

Avista Spectre/XL determines the linear frequency response of a circuit by performing a SPICE-type AC small-signal calculation. This analysis linearizes any nonlinear devices in the circuit, at their DC bias point, and evaluates the state of the circuit with all small-signal sources set to the same frequency.

Nonlinear Time Domain (HB) Results

Avista Spectre/XL determines the spectral response (steady-state operation, where time $\rightarrow \infty$) of a circuit by a harmonic balance (HB) method. This dynamic evaluation includes the nonlinear behavior of devices and signal sources. Unlike SPICE-type simulators, *Avista Spectre/XL* does not have to continue calculation until the start-up transient vanishes. The advantage of this technique, for example, is the fast and accurate evaluation of high Q and narrow band circuits.

Nonlinear Frequency-Translation (HAC) Results

Spectre/XL for RF & Microwave Design calculates the intermodulation products of mixers and receivers using “large-signal/small-signal” analysis. This frequency translation analysis uses harmonic balance to linearize devices about their time-domain (steady-state) operating point, then evaluates circuit signals in the frequency domain. Signal “up-conversion” and “down-conversion” are both supported.

Linear Noise Results

Avista Spectre/XL determines the noise at any node in your circuit, using a linear SPICE-type noise analysis. It can also determine individual device noise contributions, and find which devices are the dominant noise sources.

Frequency-Converted Noise (HNOISE) Results

Spectre/XL for RF & Microwave Design’s unique “large-signal/small-signal” analysis calculates conversion noise in mixers and phase noise in oscillators. The “Top 10” list provides instant identification of dominant noise sources. Noise “up-conversion” and “down-conversion” are both supported.

S-Parameter (SP) Results

Avista Spectre/XL determines the scattering parameters (*s*-parameters) for multi-port circuits. For two-port amplifiers, *Avista Spectre/XL* calculates stability and gain factors, and Smith chart locations for stability and gain circles. *S*-parameter results complement the N-port device that characterizes a multi-port block using *s*-parameter data.

Suitable Nonlinear Circuits

Audio and Radio Frequency Circuits

You can find signal distortion at any frequency with *Avista Spectre/XL*. You can easily determine nonlinear gain, crossover and phase distortion effects in amplifiers, filters, and other signal processing circuits. *Avista Spectre/XL* easily handles precision circuits with nonlinearities that produce distortion much less than the intended signal, which are tough to analyze with time-domain simulators. *Avista Spectre/XL* also accurately evaluates the effects of feedback, feed-forward, multi-path and parasitic circuitry.

High Frequency and Microwave Circuits

You can also use *Avista Spectre/XL* with circuits that operate above radio frequencies. Besides the design difficulties faced by lower frequency circuits, these circuits often have distributed elements, such as transmission lines, which model loss, dispersion and coupling effects. Time domain evaluation (as in SPICE) of these circuits becomes impractical when the partial differential equations that describe these elements are very complex or do not have solutions. *Avista Spectre/XL* avoids these difficulties by using the frequency domain description of these circuit elements, which you can measure with a network analyzer. *Avista Spectre/XL* can provide accurate results quickly.

Oscillator Circuits

Avista Spectre/XL finds the exact frequency of oscillation for oscillator circuits by combining the harmonic balance method with Excel's Solver analysis tool. By using simple macros, you can see how changes in control signals, component values and temperature affect oscillator operation.

Limits on Nonlinear Circuit Types

Currently, *Avista Spectre/XL* can calculate results for forced periodic systems, which include circuits such as amplifiers, filters, attenuators, and detectors. *Avista Spectre/XL* supports the analysis of self-starting circuits such as oscillators. *Avista Spectre/XL* is ideal for those circuits that have a mix of long and short time-constants. *Spectre/XL for RF & Microwave Design* calculates the frequency-translated response of multipliers, mixers, and receivers.

Avista Spectre/XL does not analyze the nonlinear behavior of switching, mixed analog/digital or digital circuitry. Analysis of these kinds of circuits will be added in a future release.

C H A P T E R 3

Demonstration Circuits

We are assuming you are familiar with *Microsoft Excel* (hereafter written as *Excel*) and that you have the following basic skills:

- Opening a workbook
- Awareness of *Excel* screen features
- Entering and editing data
- Building formulas to calculate values
- Saving a workbook

In this chapter we will open example workbooks each with an *Avista Spectre/XL* (hereafter written as *Spectre/XL*) circuit object (circuit description) embedded in it. Such a worksheet will be called a *Spectre/XL* worksheet. A *Spectre/XL* worksheet can have other objects such as Picture or *Microsoft Word* objects embedded as well.

It's a Candy-Mint, It's a Breath-Mint, It's Spectre/XL!

Spectre/XL circuit objects present two different ways of working with you and *Excel*.

- The first way is when you enter or edit the circuit description. Support of Include sheets, *s*-parameter descriptions, and context-sensitive help are some of the features provided to assist in the construction of your circuits. In the demonstration version of *Spectre/XL* you can view circuit descriptions but cannot change or create them.
- The second way is after the description has been entered and evaluation of your design is carried out. Cells on your worksheet can control the values of devices and parameters in the circuit, and *Spectre/XL* worksheet functions return circuit responses (numbers).

  When you choose a cell on your worksheet to be a control the following dialog box appears. You select which device and parameter will be controlled by that cell. This feature is not available in the demonstration version, since controls have been proved for you.

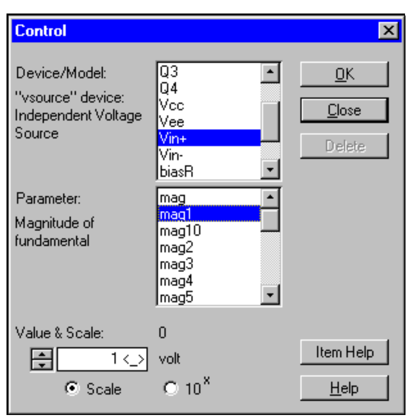


Figure 3-1 Example Control Dialog Box.

From the designer's point-of-view, *Spectre/XL* is a "circuit editor" when viewing or editing a description. However, when *Excel* calculates the contents of cells, it sees *Spectre/XL* as set of worksheet functions, and links (Controls) between worksheet cells and the circuit.

What's All This Embedding Stuff, Anyhow?

Spectre/XL takes advantage of the embedding features in Windows software such as *Excel*. *Embedding* means "to insert an information object"—such as a *Spectre/XL* circuit description or a word processor memo—into an *Excel* worksheet. All the data associated with an embedded object is stored in the worksheet. This makes it simple to transfer or distribute the work done in *Excel* to other computers, since only a single *Excel* .XLS file needs to be copied.

In-Place Activation

The *Spectre/XL* circuit object remains in place when you double-click on it for editing. The editor is *active* and used *in-place* in *Excel*. You still see the surrounding worksheet even though the *Excel* menus and toolbars have changed to those for the *Spectre/XL* circuit editor. The *Microsoft Windows* jargon for this feature is *in-place activation.*

Activation Changes Excel Window

The following changes take place in the *Excel* window when a *Spectre/XL* circuit object is selected.

- A hatched border appears, surrounding the circuit description.
- The name in the window title bar changes from "Excel - *workbook.xls*" to "Excel - Spectre/XL Circuit in *workbook.xls*."
- All the menus except the **File** and **Window** menus change to those for the *Spectre/XL* circuit editor.
- The toolbars change to the *Spectre/XL* circuit editor toolbar.

You return to *Excel* by clicking anywhere outside the *Spectre/XL* circuit object or by pressing the ESC key.

A Sample Circuit

We will open an example workbook that has diode mixer sample circuit to demonstrate the nonlinear analysis of *Avista Spectre/XL*.

Note Always choose **NO** when *Excel* asks to re-establish links when you open the example workbooks.

► **To open the diode mixer example in Excel**

Start *Excel*. The *Excel* window appears. If necessary, enlarge the window by clicking the Maximize button in the upper-right corner of the window.

1. Click the **Open** button.
2. Navigate to the EXAMPLES\SPXL directory which is a subdirectory of where *Excel* is installed (usually \MSOFFICE\EXCEL).
3. EXCEL 7.0: In the **Listed Files** box, select RFdemo_1.XLS. Click the **Open** button.

   EXCEL 5.0: In the **File Name** box, select RFdemo_1.XLS. Click the **OK** button.

4. *Excel* will ask whether to re-establish links as shown in Figure 3-2. Choose **NO**.
5. Wait for a few moments for the worksheets in the workbook to be recalculated. There are three worksheets labeled “About Demo”, “Diode mixer”, and “Functions”.

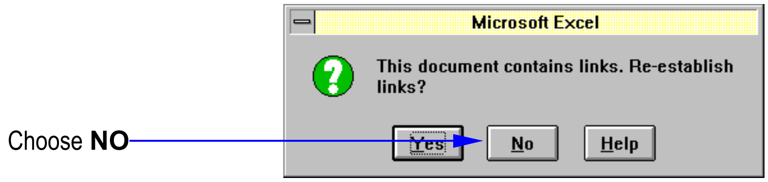


Figure 3-2 Re-Establish Links Dialog Box.

Viewing the Sample Circuit

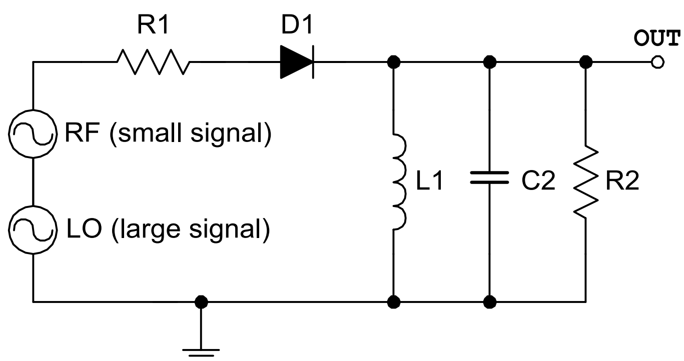


Figure 3-3 Diode Mixer Schematic.

The sample circuit will open to a worksheet that calculates the results. To view a discussion of the mixer example, choose the tab labeled “About Demo” at the bottom of the sheet. At the top of the diode mixer worksheet, you will see several lines of text bounded by a border. This is the embedded *Spectre/XL* circuit description. Double-click on it to *activate* the circuit editor to view the circuit *in-place*.

The circuit editor is very similar to the Notepad text file editor that is supplied with *Microsoft Windows*. You can perform several kinds of edit operations on text by using the mouse or keyboard. These operations include copying, cutting, clearing, and pasting. Because, the circuit editor uses the Windows Clipboard, text can be moved between the circuit editor and other applications such as a word processor.

In the demonstration version, the circuit editor acts as a circuit viewer only. You cannot make any changes to the circuit description in the demonstration version.

Evaluating the Sample Circuit

Click outside the circuit description somewhere on the spreadsheet to return to *Excel*. Follow the hints shown on the worksheet for changing the values of the two signal frequencies and signal levels outlined in blue. Every time you change a value the circuit is recalculated. Use the F9 key to recalculate a table if shown.

Worksheet Functions

Every piece of electrical information is available through the use of worksheet functions. You can use the *Excel* Function Wizard to place *Spectre/XL* specific worksheet functions in cells and display the electrical properties of the sample circuit.

See the next chapter, “Using Spectre/XL,” for more information on *Spectre/XL* functions.

► **To Insert a Spectre/XL worksheet function in a cell**

1. Select an empty cell.
2. Choose **Function** from the **Insert** menu, or select the **Function Wizard** from the Standard toolbar.
3. Scroll the **Function Category** box and select **Spectre/XL**.
4. Select **DC** from the **Function Name** box.
5. Select the **Next** button to get to the next step of the **Function Wizard**.
6. Enter `lo` for the **Item Name** and press the **Tab** key. Since node names and parameters in circuit descriptions are case sensitive i.e. `LO` is different from `lo`, be sure to use the correct identifier.
7. Select **Finish**.

The value of the voltage at node `lo` in the circuit description is displayed in the selected cell.

Quick Summary

This following summary highlights what you have learned:

- Double-click on a *Spectre/XL* circuit object to view the circuit description.
- The circuit editor remains active until you click outside the circuit object or by pressing the ESC key.
- Identifiers and parameters in circuit descriptions are case sensitive.
- Controls link cells on the worksheet with parameters in the circuit description.
- Worksheet functions return the electrical properties of the circuit.

A Linear Amplifier Circuit

The description for a linear amplifier has been entered in the workbook RFdemo_2.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. See Figure 3-4 for a schematic of the circuit. This workbook uses *Excel*’s optimizer to flatten the gain and minimize input and output reflections.

► **To open the linear amplifier example in Excel**

1. Click the **Open** button.
2. Navigate to the EXAMPLES\SPXL directory which is a subdirectory of where *Excel* is installed (usually \MSOFFICE\EXCEL).
3. <u>EXCEL 7.0</u>: In the **Listed Files** box, select RFdemo_2.XLS. Click the **Open** button.

   <u>EXCEL 5.0</u>: In the **File Name** box, select RFdemo_2.XLS. Click the **OK** button.
4. *Excel* will ask whether to re-establish links as shown in Figure 3-1. Choose **NO**.
5. Wait several moments for the worksheets in the workbook to be recalculated.
   You will see a worksheet that describes the example. Click on the tab labeled “Differential-Pair Amplifier” to see the calculations and results.

Follow the instructions on the screen to optimize the performance of the amplifier. This may take several minutes to recalculate.

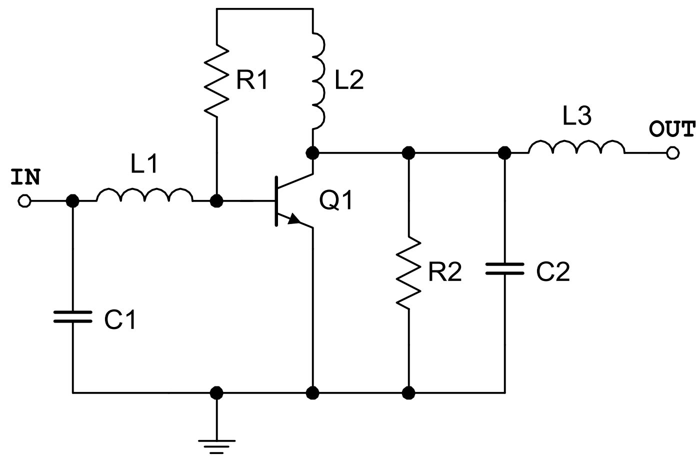


Figure 3-4 Linear Amplifier Schematic.

Other Circuits

Analysis of conversion noise of a mixer is illustrated in the workbook RFdemo_3.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. This workbook plots both linear and converted noise versus frequency.

The description for a differential amplifier has been entered in the workbook DEMO_2.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. See Figure 3-5 for a schematic of the circuit. This workbook has 3-D plots showing the trade-off of large-signal gain and distortion versus emitter and base resistance.

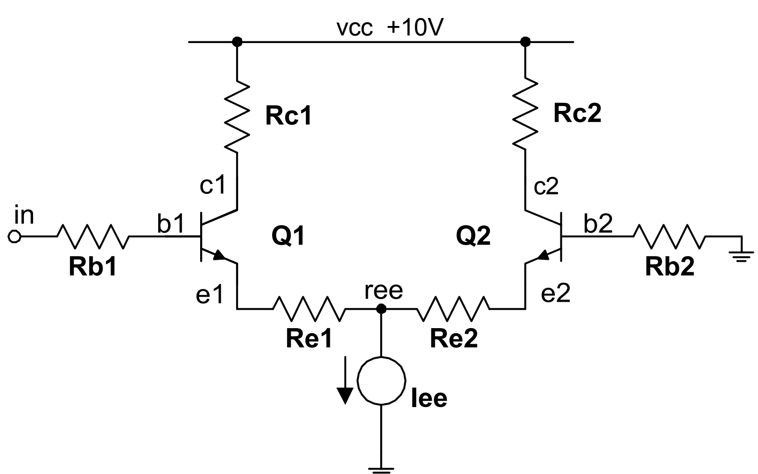


Figure 3-5 Differential-Pair Amplifier Schematic.

The description for a 5 pole Chebychev active filter with Bode plots has been entered in the workbook DEMO_3.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. See Figure 3-6 for a schematic of the circuit.

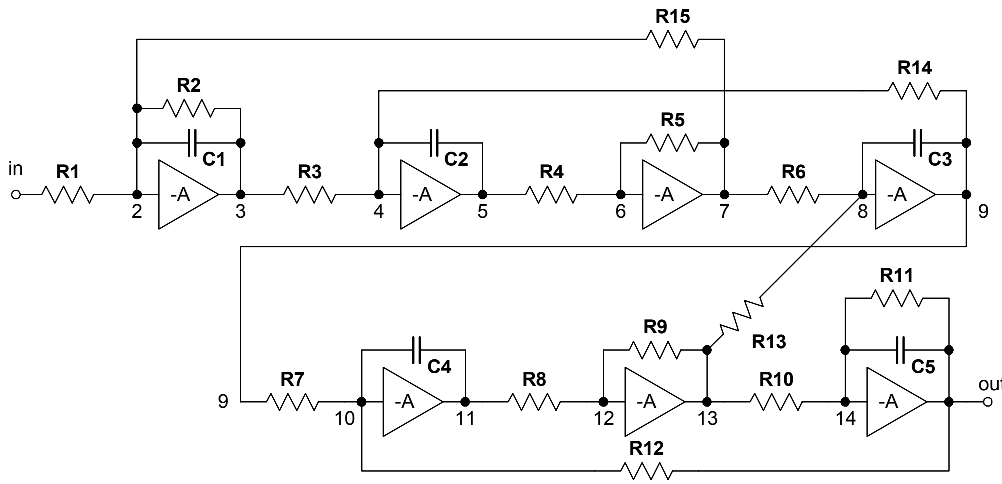


Figure 3-6 Chebychev Filter Schematic.

The description for a totem-pole output circuit (TTL gate) showing "DC sweep" charting has been entered in the workbook DEMO_4.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. This example uses Excel's "Goal Seek" tool for setting circuit values. See Figure 3-7 for a schematic of the circuit.

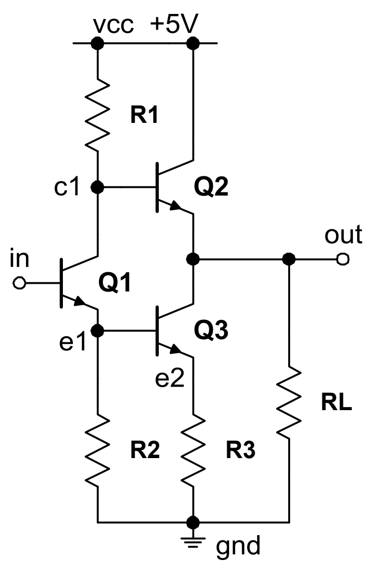


Figure 3-7 Totem-Pole Output Schematic.

The description for a common-emitter amplifier has been entered in the workbook DEMO_5.XLS which is found in the EXCEL\EXAMPLES\SPXL directory. This example use *Excel*'s Solver to "optimize" an amplifier to satisfy design requirements. See Figure 3-8 for a schematic of the circuit.

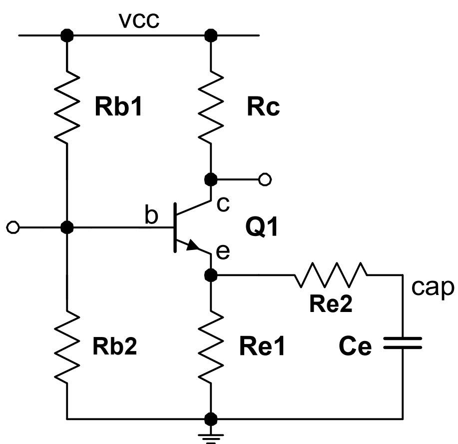


Figure 3-8 Common-Emitter Amplifier Schematic.

The example DEMO_6.XLS shows s-parameter extraction and calculations for a 2-port amplifier circuit, including stability and gain circle locations on a Smith chart. The circuit is a single n-port device. This example is derived from Maas, *Nonlinear Microwave Circuits*, Prentice Hall 1991.

The example DEMO_7.XLS shows the noise contributions of several components in a BJT common-emitter amplifier across a band of frequencies. Also shown is an ordered list of devices that are the significant noise contributors.

This page is intentionally left blank.

C H A P T E R 4

Using Avista Spectre/XL

Comparing Spectre/XL to SPICE

Sweeping

In SPICE, sweeping is a part of the .DC statements. In many commercial versions, a .STEP statement is also included for varying a component across multiple small-signal (`.AC`) and transient (`.TRAN`) runs as well. Since sweeping and circuit measurement are combined into one operation or statement, this restricts the kind and number of sweeps available.

Spectre/XL separates the sweep feature from its function evaluations. Multiple parameters of any type can be varied from the worksheet to investigate performance tradeoffs. For example, it is very simple to sweep two resistors, independently, and chart the AC characteristics of a circuit across the range of values in *Spectre/XL*. This would be difficult to impossible in most versions of SPICE without creating custom post-processing routines. *Spectre/XL* is "policy-free" when it comes to getting the desired information from your circuit.

Charting and Plotting

Spectre/XL is unique in its ability to chart 3-D surface plots of data tables. For example, charting a measurement versus varying two component parameters is simple to do. Since evaluation results can be manipulated and analyzed on the worksheet, charts can show not only electrical quantities such as voltage or power, but circuit specifications such as gain, and bandwidth.

Graphing of DC and other single data-point results is also supported. Most SPICE-type graphing programs only allow sets or curves of data to be plotted.

Excel supports over 100 kinds of charts, providing the richest selection available.

Frequency Domain

Spectre/XL makes the measurement at the exact frequency specified, including DC (0 hertz). Most SPICE-type simulators do not allow AC runs at a single frequency. You do not have to inspect an output (.OUT) file or read an interpolated result from a plot.

Excel's Goal Seek can find the exact frequency for you, given a response goal. An example is searching for a -3dB response by adjusting the frequency for an AC function. To find the exact frequency for peaks and nulls in a specified range, you can use *Excel*'s Solver.

Time Domain

SPICE determines time-domain response by linear approximations that step forward, point by point, in time. Nonlinear behavior is evaluated only at these points and the energy stored in capacitors and inductors is integrated between the points. Needless to say, SPICE must start from "time = 0" and work forward. If you are interested in the steady-state operation of an oscillator, SPICE must be used carefully and the full start-up of the oscillator must be simulated. If you are looking for distortion products way below the main signal, SPICE's piecewise-linear method is working against you.

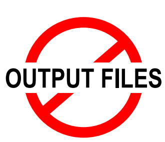
Spectre/XL determines time-domain response, for systems with an input signal, by harmonic analysis. This analysis uses the fact that every time varying signal has an equivalent frequency-domain representation. *Spectre/XL* uses the *harmonic balance* method to determine the signals in the circuit, up to a frequency limit that is automatically adjusted to ensure accurate results. This approach makes *Spectre/XL* ideal for analyzing low-distortion circuits, oscillators, and high-frequency circuits that often are not (or cannot) be modeled in SPICE. *Spectre/XL* is fast for finding steady-state results since it works without starting from an initial condition (time = 0).

Using harmonics (Fourier coefficients) to represent signals is not as limited as it might sound. Just seven harmonics will provide a surprisingly accurate representation for a halfwave-rectified sinewave, and it will be far more accurate than you would probably use doing "back of the envelope" calculations for a circuit. However, *Spectre/XL* is not able, in this release, to simulate start-up transients, or the evolution of signals that don't repeat.

One aspect of RF and microwave circuits is using mixer circuitry for frequency conversion. Mixers convert (heterodyne) one input frequency to another frequency, where filtering and amplification are easier to implement. Because of the disparate frequencies involved this is a challenging problem for a circuit simulator. SPICE-type transient analysis must simulate every cycle of the highest frequency waveform in the circuit, for as many cycles as is needed to extract an accurate result. Single-tone harmonic balance will work only if the signals being mixed are commensurate frequencies (their difference frequency is a common divisor for both signals). However, the number of harmonics required to include all intermodulation products will probably swamp the simulator. Multi-tone harmonic balance is effective but at the cost of calculating and storing data from the multidimensional Fourier transforms required for each nonlinear device in the circuit.

Because communications signals are generally weak, linear analysis is widely used for manual calculation and computer simulation of filters and amplifiers. The signals are clearly weak enough that they will not invoke a significant nonlinear response from the circuit, such as changing its bias point or driving it into saturation. Linear simulators, such as Touchstone™, must be given the linear information (such as hybrid-pi values for bipolar transistors, or s-parameter tables) for nonlinear devices in the circuit. Nonlinear simulators, such as SPICE and Libra, can develop linear information "on the fly" by finding a DC bias-point and calculating the linear equivalent for each nonlinear device. This is how a single, nonlinear Gummel-Poon (bipolar transistor) model can produce an accurate, linear hybrid-pi model for wide variety of bias situations. After the linear equivalent is known, the simulator can proceed with small-signal (AC) analysis. However, it is up to the engineer to determine that the signal levels calculated in linear analysis do not invoke a nonlinear response that would lead to inaccurate results.

Spectre/XL provides linear simulation for frequency-conversion circuits, such as mixers, by applying the "bias-point/small-signal" technique SPICE and Libra use to develop a linear equivalent circuit from a nonlinear bias-point. Instead of using a nonlinear DC bias-point, *Spectre/XL* uses a nonlinear harmonic balance solution as a starting point for small-signal analysis. This "large-signal/small-signal" technique provides quick and accurate results.



Noise

SPICE provides the .NOISE statement to report the linear noise in your circuit but allows only one .NOISE statement per simulation. To consider another output, you are required to run the simulation again. Also, you are required to view the SPICE output (.OUT) file to manually determine which devices are the largest noise contributors. Unfortunately, the list of noise sources is not sorted by size of contribution; this task is left to the user. *Spectre/XL* conveniently reports the noise at any node in your circuit and maintains a sorted list of noise contributors. SPICE cannot analyze the frequency translation of noise in a circuit. Frequency-translated noise and phase noise analysis is available in only *Spectre/XL for RF & Microwave Design*.

S-Parameter and Two-Port Analysis

Berkeley SPICE does not provide *s*-parameter or two-port analysis of a circuit. Scattering parameters (*s*-parameters) tell you “everything you need to know” about small-signal amplifier design except for noise. This analysis treats a circuit as a “black box” with multiple ports. Using *s*-parameters, impedance matching, gain, input and output VSWR, and stability can be expressed in equations . Two-port analysis is a convenient method of finding an amplifier's stability and gain over changes in frequency, and input and output loading.

When Should I Use SPICE?

You should use SPICE when you need to verify the operation of circuits with signals that are piece-wise linear (PWL) and non-repeating. In these situations there is no single steady-state solution. How the circuit operates during the transition from one state to another is what needs to be determined. SPICE transient analysis should be used to show this behavior. Digital logic is typical example of this circuit type.

If the response of a circuit to a square wave needs to determined in *Spectre/XL*, you can approximate the shape of a square wave by specifying a source that uses all 10 possible harmonics.

Using Existing SPICE Netlists

Spectre/XL support the import of SPICE netlists via the Paste Special command on the Edit menu in the circuit editor. The Paste Special command gives you the choice of a translation from SPICE2 or SPICE3 syntax. This command is not available in the demonstration version.

Comparing Spectre/XL to Libra and Touchstone

Sweeping

In Libra and Touchstone, sweeping is a part of the FREQ block. Since sweeping and circuit measurement are combined into one block, this restricts the kind and number of sweeps available.

Spectre/XL separates the sweep feature from its function evaluations. Multiple parameters of any type can be varied from the worksheet to investigate performance tradeoffs. For example, it is very simple to sweep two resistors, independently, and chart the characteristics of a circuit across the range of values in *Spectre/XL*. *Spectre/XL* is “policy-free” when it comes to getting the desired information from your circuit.

Charting and Plotting

Spectre/XL is unique in its ability to chart 3-D surface plots of data tables. For example, charting a measurement versus varying two component parameters is simple to do. Since evaluation results can be manipulated and analyzed on the worksheet, charts can be adjusted without revisiting the circuit netlist.

Excel supports over 100 kinds of charts, providing the richest selection available.

Get the Result You Need

Spectre/XL makes the measurement at the exact frequency specified, including DC (0 hertz). You do not struggle with the simulator to get the result you need. You do not have to inspect an SCN file or read an interpolated result from a plot.

Excel’s Goal Seek can find the exact frequency for you, given a response goal. An example is searching for a -3dB response by adjusting the frequency for an AC function. To find the exact frequency for peaks and nulls in a specified range, you can use *Excel*'s Solver. Spectre/XL is not limited to setting frequency or power but can set exact values for any component of your design.

Temperature dependent effects in semiconductors are simulated in Spectre/XL but not in Libra nor Touchstone.

Nonlinear

Spectre/XL determines time-domain response, for systems with an input signal, by harmonic analysis. This analysis uses the fact that every time varying signal has an equivalent frequency-domain representation. *Spectre/XL* uses the *harmonic balance* method to determine the signals in the circuit, up to a frequency limit that is automatically adjusted to ensure accurate results. This approach makes *Spectre/XL* ideal for analyzing low-distortion circuits, oscillators, and high-frequency circuits that often are not (or cannot) be modeled in SPICE. *Spectre/XL* is fast for finding steady-state results since it works without starting from an initial condition (time = 0).

One aspect of RF and microwave circuits is mixer circuitry for frequency conversion. Mixers convert (heterodyne) one input frequency to another frequency, where filtering and amplification are easier to implement.

Spectre/XL provides linear simulation for frequency-conversion circuits, such as mixers, by applying the “bias-point/small-signal” technique SPICE and Libra use to develop a linear equivalent circuit from a nonlinear bias-point. Instead of using a nonlinear DC bias-point, *Spectre/XL* uses a nonlinear harmonic balance solution as a starting point for small-signal analysis. This “large-signal/small-signal” technique provides quick and accurate results.

Noise

Libra and Touchstone provide noise analysis for linear noise in your circuit. You are limited in the kinds of noise information that can be found. There is no easy means to find out what are the largest noise contributors. *Spectre/XL* conveniently reports the noise at any node in your circuit and maintains a sorted list of noise contributors. Neither Libra nor Touchstone can analyze the frequency translation of noise in a circuit. Frequency-translated noise and phase noise analysis is available in only *Spectre/XL for RF & Microwave Design*.

Worksheet Functions

Spectre/XL provides worksheet functions to deliver the results from circuit evaluations. They are used in the same way as other *Excel* functions. Arguments to the functions in **bold** type are required and the remaining arguments in regular type are optional.

Note Each of the arguments to the functions below can be a text string or number, a reference to a cell containing a text string or a number, or a macro returning a text string or a number.

AC for Small-Signal Response

The **AC** function returns a small-signal response (voltage) as a 1-by-2 array consisting of the magnitude and phase respectively.

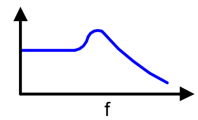
Definition **AC (Item, FreqHz)**

Item is the net name for the voltage to return.

FreqHz is the frequency at which the response is evaluated.

The AC small-signal portion of *Spectre/XL* computes the result in phasor form.

If the circuit has only one AC source, it is often convenient to set its magnitude to unity and its phase to zero. In this way the small-signal transfer function is directly computed. Some example uses are:



- Determining frequency response of amplifiers and filters.
- Determining phase response and group delay.
- Determining input and output impedance.
- Determining unstable loop gain.

AC Examples

In the first example, the value of the node `Output` at a frequency of 100 megahertz is returned. In the second example, if the worksheet cell J14 contains the string Output and the worksheet cell K14 contains the number 100E+6 then the first two examples return the same result. The third example returns the AC response for `Output1` at 60 hertz. The fourth example returns the response of the node `out` normalized to the DC response of node `out`.

```
=AC("Output", 100E+6)
=AC(J14, K14)
=AC(Concatenate("Output",1),60)
```

DC for DC Bias-Point

The **DC** function returns a DC operating value (voltage).

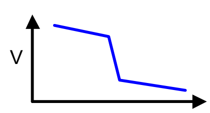
Definition

DC (**Item**)

Item is the net name of the voltage to return.

DC evaluation determines the bias-point of the circuit with inductors shorted, capacitors opened, and all time varying sources set to their quiescent value.

Some example uses are:



- Setting DC bias point or finding bias point sensitivity to component changes.
- Determining quiescent power dissipation of devices.
- Determining small-signal gain versus DC input.
- Determining a transfer function or DC load line for an amplifier.

DC Examples

In the first example, the voltage at node `Vbase` is returned. If the worksheet cell A9 contains the string `Vbase,` then the second example performs the same function as the first. The third example returns the voltage at node 6.

```
=DC("Vbase")
=DC(A9)
=DC(2*3)
```

HAC for Heterodyned AC (Frequency-Converted Small-Signal Response)

The **HAC** function returns the intermodulation products (voltage) due to a periodic stimulus, as a 1-by-2 array consisting of the magnitude and phase respectively.

Definition

HAC (**Item**, **FundHz**, **Harm, FreqHz**)

Item is the net name of the voltage to return.

FundHz is the frequency for the large-signal source(s).

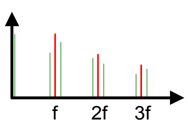
Harm is the harmonic around which intermodulation products will be calculated. Specifying 1 calculates intermodulation products around the fundamental of the large-signal source. Specifying 0 calculates products around DC (0 Hz).

FreqHz is the frequency for the small-signal source(s). It is also the offset frequency from the harmonic at which an intermodulation product will be calculated.

The HAC evaluation computes the heterodyne (frequency conversion) response of a circuit. The large-signal source establishes the frequency conversion of the circuit. The small-signal source stimulates the circuit to produce intermodulation products. For a complete discussion see Heterodyne AC (HAC) in the Chapter 7, "Advanced Topics."

Some example uses are:

- Determining the intermodulation distortion of an amplifier.
- Determining conversion gain of a mixer.
- Determining optimal local oscillator (LO) power for a mixer.
- Determining frequency and harmonic content of a converter.



Some example uses are:

- Determining the intermodulation distortion of an amplifier.
- Determining conversion gain of a mixer.
- Determining optimal local oscillator (LO) power for a mixer.
- Determining frequency and harmonic content of a converter.

HAC Examples

In the first example, the magnitude and phase of the fifth harmonic of node `Output` is returned in an array based on a large-signal stimulus at 1 MHz, and a small-signal stimulus of 440 Hz. In the second example, if the worksheet cells A4, B4, C4, D4 have the same arguments as the previous example, the same function will be performed. The third example finds the nth odd harmonic of node `50` at a large-signal frequency specified by `CARRIER` and the small-signal frequency specified by the expression `2*BASEFRQ`.

```
=HAC("Output", 1E6, 5, 440)
=HAC(A4, B4, C4, D4)
=HAC(50, CARRIERFRQ, 2*N-1, 2*BASEFRQ)
```

HB for Time Domain Response

The **HB** function returns the periodic stead-state response (voltage) to a periodic stimulus as a 1-by-2 array consisting of the magnitude and phase respectively.

Definition

HB (**Item**, **FundHz**, **Harm**)

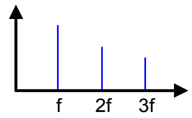
Item is the net name of the voltage to return.

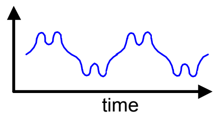
FundHz is the fundamental frequency on which all responses are based.

Harm is the number of the harmonic that will be calculated. Specifying 0 calculates the average DC level. Specifying 1 calculates the fundamental.

The HB evaluation computes the periodic steady-state response of a circuit.

Some example uses are:





- Determining the linearity of an amplifier.
- Determining how component changes affect the large-signal performance.
- Determining maximum input drive level for a filter.
- Determining frequency and harmonic content of an oscillator.
- Verify correct large-signal operation when optimizing DC and AC characteristics such as power consumption and bandwidth.

HB Examples

In the first example, the magnitude and phase of the seventh harmonic of node `Output` is returned in an array based on a fundamental of 1 MHz. In the second example, if the worksheet cells A4, B4, C4 have the same arguments as the previous example, the same function will be performed. The third example finds the nth odd harmonic of node `5` at a frequency specified by the expression `2*BASEFRQ`.

```
=HB("Output", 1E6, 7)
=HB(A4, B4, C4)
=HB(5, 2*BASEFRQ, 2*N-1)
```

HNOISE for Heterodyned Noise (Frequency-Converted Noise)

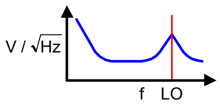
The **HNOISE** function returns the heterodyned (frequency converted) spectral noise density, in volt / $\sqrt{\text{Hz}}$, at the specified net name and frequency, or returns a text string containing the device name for the ***DevNum***th greatest noise contributor, or returns the noise contribution of the device specified in **DevName**. For unconverted linear noise see **NOISE**.

Definition

HNOISE (**Item**, **FundHz**, **Harm**, **FreqHz**, {DevNum | DevName})

Item	is the net name of the noise voltage to return.
FundHz	is the fundamental frequency of the large-signal source(s).
Harm	is the number of the harmonic around which the noise voltage is calculated. Specifying 1 calculates noise around the fundamental of the large-signal source. Specifying 0 calculates noise around DC (0 Hz).
FreqHz	is the offset frequency from the harmonic at which the noise voltage is calculated.
DevNum	(optional) is the index into the list of the "Top 10" noise contributors. Specifying 1 causes **HNOISE** to return the name of the largest noise contributor. Specifying an index greater than the number of noise sources, or 10, causes **HNOISE** to return an *Excel* `#N/A` value.
DevName	(optional) is the name of the device whose noise contribution is calculated. Specifying a device that is not in the circuit causes **HNOISE** to return an *Excel* `#N/A` value.

Noise is produced by ports, resistors, and semiconductor devices. Heterodyned (frequency converted) noise results from noise passing through a frequency-conversion circuit such as a mixer or multiplier, and other circuits that operate nonlinearly. The noise translation behavior of a circuit is established by the large-signal sources in the circuit at the frequency specified by **FundHz**. Noise results are calculated around the fundamental, or at one of its harmonics based on the value of **Harm**. Choosing a value of 0 for **Harm** calculates noise around DC (0 Hz). **FreqHz** specifies the frequency offset from the harmonic at which the noise voltage is calculated. For a complete discussion see Heterodyne Noise (HNOISE) in Chapter 7, "Advanced Topics."



Noise is produced by ports, resistors, and semiconductor devices. Some example uses are:

- Determining the phase noise of an oscillator.
- Determining the conversion noise of a mixer.
- Determining which components are the dominant noise sources.

HNOISE Examples

In the first example, the heterodyned noise for the node `Output` at 1.05 MHz is calculated around a 1 MHz fundamental. In the second example, if the worksheet cells A4, B4, C4, D4 have the same arguments as the previous example, the same calculation will be performed. The third example finds the nth highest heterodyned noise contributor to node `5` at a frequency of 100 Hz, based on a fundamental specified by the expression `2.0*SRCFRQ`. The fourth example returns the noise contribution of the largest source to node `Out` around the fundamental frequency specified by `Carrier`.

```
=HNOISE("Output", 1E6, 1, 1.05E6)
=HNOISE(A4, B4, C4, D4)
=HNOISE(5, 2.0*SRCFRQ, 0, 100, n)
=HNOISE("Out", Carrier, 1, Freq, HNOISE("Out",Carrier,1,Freq, 1))
```

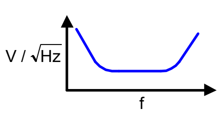
NOISE for Linear Noise

The **NOISE** function returns the noise voltage, in volt / $\sqrt{\text{Hz}}$ at the specified net name and frequency, or returns a text string containing the device name for the ***DevNum***th greatest noise contributor, or returns the noise contribution of the device specified in **DevName**.

Definition

NOISE (**Item**, **FreqHz**, {DevNum | DevName})

Item	is the net name of the noise voltage to return.
FreqHz	is the frequency at which the response is evaluated.
DevNum	(optional) is the index into the list of the “Top Ten” noise contributors. Specifying 1 causes NOISE to return the name of the largest noise contributor. Specifying an index greater than the number of noise sources, or 10, causes NOISE to return an *Excel* `#N/A` value.
DevName	(optional) is the name of the device whose noise contribution is calculated. Specifying a device that is not in the circuit causes NOISE to return an *Excel* `#N/A` value.



Noise is produced by ports, resistors, and semiconductor devices. Some example uses are:

- Determining the noise of an amplifier.
- Determining which components are the dominant noise sources.

NOISE Examples

In the first example, the noise for the node `Output` at 1 MHz. In the second example, if the worksheet cells A4, B4 have the same arguments as the previous example, the same function will be performed. The third example finds the nth highest noise contributor to node `5` at a frequency specified by the expression `2*BASEFRQ`. The fourth example returns the noise contribution of the largest source to node `Out`.

```
=NOISE("Output", 1E6)
=NOISE(A4, B4)
=NOISE(5, 2*BASEFRQ, N)
=NOISE("Out", Freq, NOISE("Out", Freq, 1))
```

SP for *S*-Parameter and Two-Port Analysis

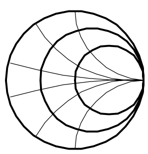
The SP function returns an *s*-parameter analysis, which is a small-signal analysis, of a circuit containing **port** devices. These devices identify the ports of a circuit, supply a reference impedance and possibly a stimulus. Besides *s*-parameters, many other values useful for analyzing amplifiers and systems are returned.

Definition **SP** (**Item**, **FreqHz**, Gain)

Item identifies the type of value to return. It is not case sensitive. Items "k" and below are for linear two-port amplifiers. Items "cl" and below are circles on a Smith chart. For those items that return a center point, the result is a 1-by-2 array (one row and two columns).

FreqHz is the frequency at which the response is evaluated.

Gain (optional) is a numeric value for the **cp**, **rp**, **ca**, **ra** analysis functions. Specify the amount of gain as a ratio (not in dB).



The SP function is a specialized form of small-signal analysis that simulates the operation of a vector network analyzer. Some example uses are:

- Determine how a circuit will function as a component in a larger system.
- Creating impedance-matching networks.

Since each "port" device has an identifying number, these are used for specify the ports when evaluating *s*-parameter values. **For the two-port amplifier analysis functions: The lower port number identifies the source (input) port and the higher port number identifies the load (output) port.**

Item	Discussion
Sij	Returns an *s*-parameter. For example "`s21`" would return the transmission coefficient from port 1 to port 2. If either port number is 10 or greater, then insert a colon between the port numbers, for example "`s13:15`".
det	Returns the determinant Δ of the *s*-matrix for a two-port circuit. $\Delta = s_{11} \cdot s_{22} - s_{12} \cdot s_{21}$
K	Returns the stability factor K defined for checking unconditional stability, for linear two-port amplifiers. $K = \frac{1 - \lvert s_{11}\rvert^2 - \lvert s_{22}\rvert^2 + \lvert\Delta\rvert^2}{2 \cdot \lvert s_{12} \cdot s_{21}\rvert}$ The necessary and sufficient conditions for unconditional stability are: $K > 1, \quad \lvert s_{11}\rvert < 1, \quad \lvert s_{22}\rvert < 1, \quad \lvert s_{12} \cdot s_{21}\rvert < 1 - \lvert s_{11}\rvert^2, \quad \lvert s_{12} \cdot s_{21}\rvert < 1 - \lvert s_{22}\rvert^2$
mu	Returns the stability parameter μ defined using a geometric approach, for linear two-port amplifiers. $\mu = \frac{1 - \lvert s_{11}\rvert^2}{\lvert s_{22} - s_{11}^* \cdot \Delta\rvert + \lvert s_{12} \cdot s_{21}\rvert}$ This parameter can replace the Linvill, Rollet, and auxiliary conditions (see K above). μ > 1 alone is necessary and sufficient for a circuit to be unconditionally stable.
msg	Returns the maximum stable gain, for linear two-port amplifiers, defining the highest available gain when the stability factor K = 1. $G_{MSG} = \left\lvert \frac{s_{21}}{s_{12}} \right\rvert$

cl Returns the center point of the load stability circle on the Smith chart, for linear two-port amplifiers.

$$C_L = \frac{s_{22}^* - s_{11} \cdot \Delta^*}{|s_{22}|^2 - |\Delta|^2}$$

rl Returns the radius of the load stability circle on the Smith chart, for linear two-port amplifiers.

$$r_L = \left| \frac{s_{12} \cdot s_{21}}{|s_{22}|^2 - |\Delta|^2} \right|$$

cs Returns the center point of the source stability circle on the Smith chart, for linear two-port amplifiers.

$$C_S = \frac{s_{11}^* - s_{22} \cdot \Delta^*}{|s_{11}|^2 - |\Delta|^2}$$

rs Returns the radius of the source stability circle on the Smith chart, for linear two-port amplifiers.

$$r_S = \left| \frac{s_{12} \cdot s_{21}}{|s_{11}|^2 - |\Delta|^2} \right|$$

cp Returns the center point of the constant power gain on the Smith chart, for linear two-port amplifiers. The optional argument of the SP function indicates the amount of *gain*, expressed as a ratio (not in dB).

$$C_p = \frac{\left(s_{22}^* - s_{11} \cdot \Delta^*\right) \cdot g_p}{1 + \left(|s_{22}|^2 - |\Delta|^2\right) \cdot g_p} \quad \text{where } g_p = \frac{gain}{|s_{21}|^2}$$

rp Returns the radius of the constant power gain on the Smith chart, for linear two-port amplifiers. The optional argument of the SP function indicates the amount of *gain*, expressed as a ratio (not in dB).

$$r_p = \frac{\sqrt{|s_{12} \cdot s_{21}|^2 \cdot g_p^2 - \left(1 - |s_{11}|^2 - |s_{22}|^2 + |\Delta|^2\right) \cdot g_p + 1}}{\left|1 + \left(|s_{22}|^2 - |\Delta|^2\right) \cdot g_p\right|} \quad \text{where } g_p = \frac{gain}{|s_{21}|^2}$$

ca Returns the center point of the constant available gain circle on the Smith chart, for linear two-port amplifiers. The optional argument of the SP function indicates the amount of *gain*, expressed as a ratio (not in dB).

$$C_a = \frac{\left(s_{11}^* - s_{22} \cdot \Delta^*\right) \cdot g_a}{1 + \left(|s_{11}|^2 - |\Delta|^2\right) \cdot g_a} \quad \text{where } g_a = \frac{gain}{|s_{21}|^2}$$

ra Returns the radius of the constant available gain circle on the Smith chart, for linear two-port amplifiers. The optional argument of the SP function indicates the amount of *gain*, expressed as a ratio (not in dB).

$$r_a = \frac{\sqrt{\left|s_{12} \cdot s_{21}\right|^2 \cdot g_a^2 - \left(1 - \left|s_{11}\right|^2 - \left|s_{22}\right|^2 + |\Delta|^2\right) \cdot g_a + 1}}{\left|1 + \left(\left|s_{11}\right|^2 - |\Delta|^2\right) \cdot g_a\right|} \quad \text{where } g_a = \frac{gain}{\left|s_{21}\right|^2}$$

SP Examples

In the first example, the *s*-parameter S21 is calculated at 1 MHz. In the second example, if the worksheet cells A4, B4 have the same arguments as the previous example, the same function is performed. The third example finds the stability parameter μ at a frequency specified by the expression `2*BASEFRQ`. The fourth example returns the radius of the constant available gain circle at a frequency specified by `Freq` and a gain of `12.5`.

```
=SP("s21", 1E6)
=SP(A4, B4)
=SP("mu", 2*BASEFRQ)
=SP("ra", Freq, 12.5)
```

LABEL for Identifying Controls

The **LABEL** function returns a text string identifying the control associated with a spreadsheet cell.

Definition

LABEL (**CellRef**, Formating)

CellRef is a reference to the cell that has a control associated with it.

Formating An optional text string that specifies the format of the text string the function returns. The default format is "`%d:%p (%s%u)`".

Item	Includes this Text
%d	Device name.
%p	Parameter name.
%s	Scale factor for cell.
%u	Parameter units.

The LABEL function makes it easy to identify those spreadsheet cells that are controls for the *Spectre/XL* circuit description. When the Formating argument is omitted from the function, the default format **"`%d:%p (%s%u)`"** is used. The easiest way to understand this format is to look at an example. Consider a cell that is controlling the **vdc** DC voltage parameter of a voltage source named `Vcc` and the scale factor for the cell is K (1E+3). The resultant text string would be the substitution for each of the items as described in the format definition. Putting it all together we would have "`Vcc:vdc (Kvolt)`".

LABEL Examples

For the examples shown, assume the worksheet cell A9 is a control that refers to the **vdc** DC voltage parameter of a voltage source named `Vcc` and the scale factor for the cell is K (1E+3). In the first example, the label string for the control associated with cell A9 is formatted using the default format "`%d:%p (%s%u)`" and has the value "`Vcc:vdc (Kvolt)`". In the second example, the returned label string contains only the device name "`Vcc`" for control A9. The third example returns the label string for control cell A9 using the format string in cell K17. In the fourth example, the label string is "`vdc of vcc`".

```
=LABEL(A9)
=LABEL(A9,"%d")
```

```
=LABEL(A9,K17)
=LABEL(A9,"%p of %d")
```

Working with Arrays

The *Spectre/XL* functions **AC**, **HB**, and possibly **SP** return two values in a 1-by-2 array (one row and two columns). Entered into a cell as a regular *cell formula*, only the first value is displayed or available in a calculation. If these array functions are entered as an *array formula*, then both values will be available. To illustrate the difference consider the following formula which normalizes the small signal gain at a frequency with respect to the gain at DC (0 hertz):

```
=AC("out", freq) / AC("out", 0)
```

and further lets assume

`AC("out", freq)` returns `{37, 90}` and
`AC("out", 0)` returns `{10, 45}`

If you had entered it as a cell formula then only the magnitude returned by each AC function is used in the division calculation. The result would be 3.7 which is the correct answer for the division of magnitudes. If you had entered it as an array formula then both the magnitude and the phase would be used in the division calculation, respectively, the result would be {3.7, 2}. Since the results from the AC function are not treated as complex numbers but just arrays, only the magnitude calculation is correct.

You can recognize an array formula by the braces ({ }) enclosing it in the formula bar. You don't type the braces yourself. *Excel* automatically puts the braces around a formula when you enter it as an array.

Entering an Array Formula

To enter an array formula, press both the CTRL and the SHIFT then the ENTER key (instead of just pressing ENTER for a cell formula). See *Excel* Help for more help with arrays.

Tips on Entering Array Formulae

You should enter an array formula in a range of cells with the same dimensions as the resulting array produced by the formula. *Excel* can then place each value in the resulting array into one cell of the array range.

- If an array formula produces an array smaller than the selected array range, *Excel* expands the resulting array to fill the range.
- If *Excel* expands an array to fill a range larger than the array formula, `#N/A` error values appear in cells for which no valid expandable value is available.
- If an array formula produces an array larger than the selected array range, the excess values do not appear on the worksheet.
- If you select any cell in a range containing an array formula, the same array formula is displayed in the formula bar.
- You cannot change or delete individual cells in the array range because all the cells share a single formula.

For more information, see the printed and online documentation for *Excel*.

Order Form

Fax to +1-916-985-6080 to order, or call 1-800-985-6080 (U.S.A. & Canada)

Guarantee: If you are not satisfied with your order, simply return it within 45 days of date of purchase for a full refund minus shipping and a $50 handling charge.

Name ____________________

Company ____________________

Address (mail stop) ____________________

Address (street) ____________________

City ____________________

State/Province ____________________

Postal Code ____________________

Country ____________________

Phone ____________________

Fax ____________________

E-mail ____________________

	Spectre/XL	*Spectre/XL for RF & Microwave Design*
Price	$1,250 †	$1,950 †
Windows 3.1, 95/NT	both included	both included
Nonlinear DC	✓	✓
AC (small-signal)	✓	✓
S-parameters	✓	✓
Noise Analysis	✓	✓
Harmonic Balance	✓	✓
Temperature	✓	✓
Interactive Analysis	✓	✓
Circuit Tuning Optimization	✓	✓
Monte Carlo Statistical Analysis	✓‡	✓‡
Oscillator Analysis	✓	✓
Nonlinear Mixer & Receiver Analysis		✓
Conversion Noise Phase Noise		✓
RF Nonlinear Model Library		✓

† Prepaid orders.
‡ $395 *Excel* add-on *Crystal Ball*, from Decisioneering.

We accept Visa, MasterCard, and American Express cards, or a check drawn on a U. S. Bank.

☐ VISA
☐ MasterCard
☐ American Express

Card Number ____________________ Exp. Date ____________

Signature ____________________ Date ____________

Quantity	Description	Price	Total
	Spectre/XL for RF & Microwave Design	1,950†	
	Spectre/XL	1,250†	
	7.25% sales tax for California only ($141.38 per $1,950; $90.63 per $1,250)		
	Shipping (no charge for prepaid orders)		
	Total		

†Prepaid orders.

phone/fax: +1–(800 or 916)–985–6080 • e-mail: avista@avista.com • 710 Figueroa Street, Folsom, California 95630