Agilent Technologies

S-Parameter Simulation

May 2003

Notice

The information contained in this document is subject to change without notice.

Agilent Technologies makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Warranty

A copy of the specific warranty terms that apply to this software product is available upon request from your Agilent Technologies representative.

Restricted Rights Legend

Agilent Technologies
395 Page Mill Road
Palo Alto, CA 94304 U.S.A.

Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries.

Microsoft®, Windows®, MS Windows®, Windows NT®, and MS-DOS® are U.S. registered trademarks of Microsoft Corporation.

Pentium® is a U.S. registered trademark of Intel Corporation.

PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated.

UNIX® is a registered trademark of the Open Group.

Contents

Chapter 1: S-Parameter Simulation

Using the S-parameter simulation component (S_Param) from the Simulation-S_Param palette enables you to:

- Obtain the scattering parameters (S-parameters) of a component, circuit, or subnetwork and convert those parameters to Y- or Z-parameters.
- Plot, for example, the variations in swept-frequency S-parameters with respect to another changing variable.
- Simulate group delay or linear noise.
- Simulate the effects of frequency conversion on small-signal S-parameters in a circuit employing a mixer. (This is also known as analyzing a frequency-translating network.)

The Simulation-S_Param palette also contains components for general simulation options and sweeps, as well as a variety of measurement components for calculating relevant measurements.

Refer to the following topics for details on S-parameter simulation:

- "Performing an S-parameter Simulation" on page 1-2 has the minimum setup requirements for an S-parameter simulation.
- "Examples (ADS only)" on page 1-3 give detailed setups for running a basic S-parameter simulation on an amplifier, as well as examples for calculating group delay, linear noise, and frequency translation.
- "S-parameter Simulation Overview" on page 1-10 is a brief description of the S-parameter simulator and some of its methods, such as group delay and frequency conversion.
- "Noise Analysis" on page 1-14 gives some of the equations and techniques that are the basis of noise calculations.
- "S-Parameters from Various Input/Output Modes" on page 1-17 describes the features available in ADS to simulate S-parameters for designs that use various input and output modes.
- "S_Param Simulation Controller" on page 1-18 provides details on the tabs and fields in the S-Parameter simulation controller.

Performing an S-parameter Simulation

Start by creating your design, then add current probes and identify the nodes from which you want to collect data

For a successful analysis, be sure to:

- Apply ports to all inputs and outputs. Use P_1Tone or P_nTone power sources to drive inputs. Terminate all other ports using port-impedance terminations (Term). Verify impedance. The Term component is found on the Simulation-S_Param palette. Power sources are on the Sources-Freq Domain palette.
- Check the Num field for each port. The S-parameter port numbers are derived from these fields. For a 2-port circuit, you would want the input labeled as Num=1 and the output as Num=2.
- Add the S-parameter component to the schematic and double-click to edit it. For a basic simulation, fill in the fields under the Frequency tab:
  - Select the Sweep type, single point, logarithmic, or linear. For a linear or logarithmic sweep, elect to define the sweep with start/stop or center/span values.
- To calculate admittance or impedance parameters, enable the options under the Parameters tab.
- Group delay calculations can be enabled from the Parameters tab.
- You can enable frequency conversion, which is useful when analyzing circuits with standard (not user-defined) behavioral mixer models. This option is under the Parameters tab.
- To calculate noise, select the Noise tab and enable *Calculate noise*. You select a node for noise calculations from the Edit list, then click Add. Use the Mode list to sort the noise contributed by individual noise sources by name or value.

For details about each field, click *Help* from the dialog box.

For more detailed descriptions of simulation setups, refer to “Examples (ADS only)” on page 1-3.

Examples (ADS only)

This section contains examples for:

- "Simulating an Amplifier" on page 1-3
- "Calculating Group Delay" on page 1-6
- "Simulating Linear Noise" on page 1-7
- "Analyzing a Frequency Translating Network" on page 1-8
- "Eliminating Unwanted Effects" on page 1-9

These examples give detailed descriptions for setting up and running S-parameter simulations.

Simulating an Amplifier

Figure 1-1 illustrates an example setup for performing a basic S-parameter simulation of an amplifier.

Note This design, *SP1.dsn*,is in the *Examples* directory under *Tutorial/SimModels_prj*. The results are in *SP1.dds*.

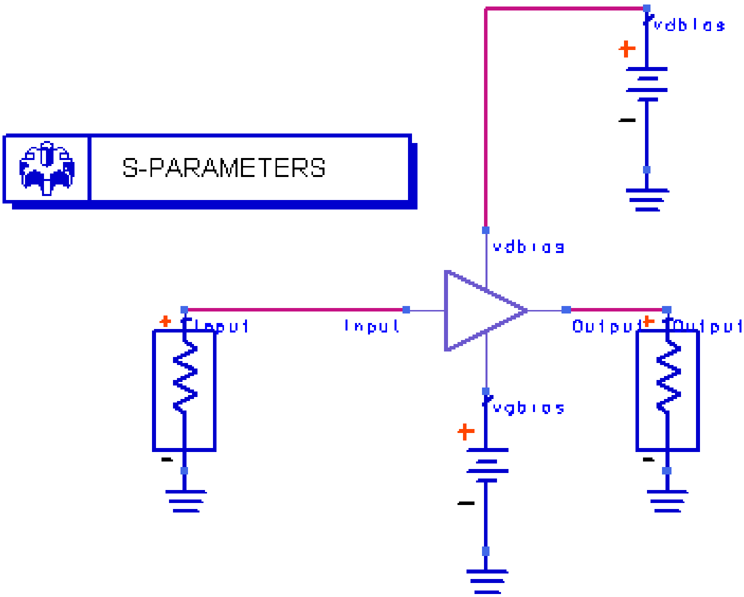


Figure 1-1. Example Setup for a Basic S-parameter Simulation

To perform a basic S-parameter simulation:

1. From the **Simulation-S_Param** palette, select a **Term** component for each port of the component or circuit to be simulated. You can edit the impedances as required, although the default value of 50 ohms is generally sufficient. Ensure that the terminations are properly connected to the component or circuit under test.

2. Ensure that the number of the input Term component is set to Num = **1**, and that of the output Term component to Num = **2**.

Note By default, the Term component provides a noise contribution (Noise = yes), but is inactive unless noise contributions are requested. Also, ensure that the number of each Term component (as defined by the component's Num parameter) is appropriate to the location of the component in the circuit, to ensure that the S-parameter data is meaningful.

---

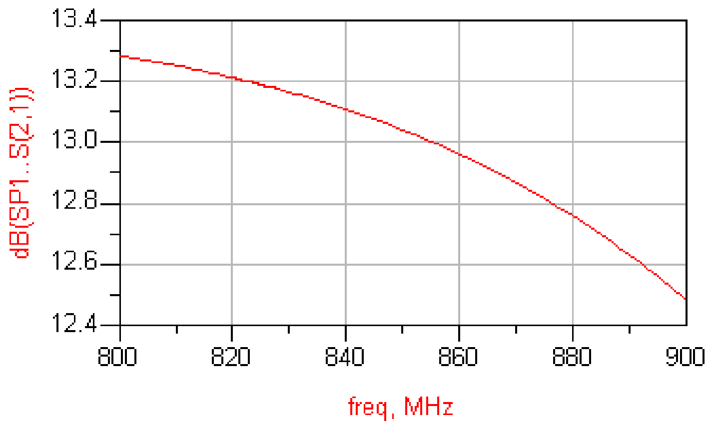
3. From the **Simulation-S_Param** palette, select **SP**. Place this component on the schematic and select the **Frequency** tab. Ensure that *Start/Stop* is selected, then set the following parameters:

    - Sweep Type = **Linear**
    - Start = **800 MHz**
    - Stop = **900 MHz**
    - Step-size = **1 MHz**

4. To obtain S-parameters, select the **Parameters** tab and ensure that *S-parameters* is selected. For a description of the options on the Parameters tab, click **Help**. To obtain Y- (admittance) and Z- (impedance) parameters, select the corresponding options.

5. Click **OK** to accept changes and close the dialog box.

6. **Simulate**. When the simulation is finished, plot S(2,1) in the Data Display. The following is a plot of the gain (S_{21}) versus frequency.



Calculating Group Delay

By measuring the transit time, with respect to frequency, of a signal through the device under test, group delay is a useful measure of phase distortion in components such as amplifiers and filters.

To calculate group delay, you enable the *Group delay* option and, if desired, set the group delay aperture. These options are under the Parameters tab. The results appear in the Data Display variables list under *delay*.

For more information, refer to "Group Delay" on page 1-12.

To calculate group delay:

1. Proceed as in "Simulating an Amplifier" on page 1-3, setting frequencies and sweep parameters as needed.
2. Edit the S-Param component, select the **Parameters** tab, and enable **Group delay**.
3. Group delay aperture is an option that is found on network analyzers and behaves similarly here. The simulator sets the frequency aperture to 0.01% of the current frequency. To override the default frequency aperture, enable **Group delay aperture** and edit the value as needed.
4. Click **OK** to accept changes and close the dialog box.
5. **Simulate**. When the simulation is finished, plot the group delay data items, identified by the prefix *delay*. This is the absolute group delay, in seconds.

Hint If the group delay data appears noisy, increase the value in the *Group delay aperture* field. If the results appear inaccurate, decrease the value. Generally, adjusting this value by a factor of 10 (in the appropriate direction) improves noisy or inaccurate results.

For an example of group delay data, refer to the topic "Obtaining Group Delay Data" in the chapter "Using Circuit Simulators for RF System Analysis" in the *Circuit Simulation* documentation.

Simulating Linear Noise

Options for simulating linear noise are available from the *Noise* tab of the S_Param simulation component. For more information about how noise is calculated, refer to “Noise Analysis” on page 1-14.

To simulate linear noise:

1. Proceed as in “Simulating an Amplifier” on page 1-3, setting frequencies and sweep parameters as needed.
2. Edit the S-parameter Simulation component and select the **Noise** tab. Then select the **Calculate noise** option.
3. In the Edit field, enter the names of the nodes at which you want noise data to be reported.

Note It is not necessary to name nodes if only noise figure is desired.

---

4. Use the Mode popup menu to sort the noise contributors (nodes) that are reported.
5. Either accept the default values for *Dynamic range to display* and *Bandwidth*, or edit these as required. The defaults are generally sufficient.
6. Click **OK** to accept changes and close the dialog box.
7. **Simulate**. When the simulation is finished, plot the noise data items. These are noise figure, identified as nf[*port_number*], and the equivalent input noise temperature, identified as te[*port_number*].

Adjusting Noise Temperature

The IEEE definition of noise figure states that it should be measured at the standard noise temperature of 290 K (16.85°C). Advanced Design System uses this definition and value of the standard noise temperature in its calculation of noise figure. For a passive circuit, if the simulation temperature is not equal to this value, the noise figure will not be the same as the loss in decibels. The simulation temperature defaults to 25°C. It can be changed by adding an Options item to the design and changing the simulation temperature there to 16.85°C.

Analyzing a Frequency Translating Network

To simulate the effects of frequency translation (also known as frequency conversion) in circuits employing mixers, the S-parameter simulator uses the same algorithm as the AC simulation component. This option causes the simulator to consider the frequency not only of the input fundamental, but also the frequency of the resulting translations. A simple model is used to calculate the reference frequencies at each node.

Selecting the *Calculate noise* option (under the *Noise* tab) will result in frequency conversion data for nonlinear noise.

For more conversion information, refer to "S-parameter Frequency Conversion" on page 1-13.

To analyze a frequency translating network:

1. Proceed as in "Simulating an Amplifier" on page 1-3.
2. Insert passive ports at locations where you want to obtain S-parameters.
3. Set frequencies and sweep parameters as needed.
4. Use a large-signal voltage or current source, such as V_1Tone or I_1Tone as the driving signal that causes the frequency translation (not a large-signal port source, such as a P_1Tone).
5. Select the **Parameters** tab, then select **Enable AC frequency conversion**.
6. In the field labeled *S-parameter freq. conv. port*, enter **1**.

Note The frequency conversion port must be the number of the input port.

7. To calculate frequency conversion data for nonlinear noise, select the **Noise** tab and enable **Calculate noise**.

Eliminating Unwanted Effects

It is sometimes helpful to reduce the contribution of other components in an analysis of a circuit involving, for example, amplifiers. The DC_Block component functions as an open during the DC part of the simulation (which is conducted automatically), while the DC_Feed component functions as an open during the S-parameter simulation. This eliminates the loss that would otherwise be experienced with the Term and the bias resistors in the circuit. Figure 1-2 illustrates the use of the DC_Block component in an example circuit.

Note This design, *Amp_wBothMatches.dsn*, is in the *Examples* directory under *MW_Ckts/LNA_prj*.

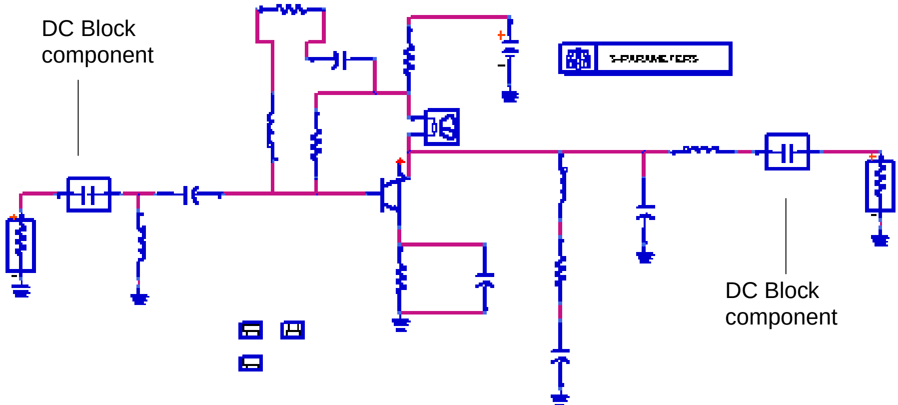


Figure 1-2. DC_Block and DC_Feed Components in a Circuit

To eliminate the effects of port and bias resistances:

1. From the **Lumped Elements** palette, select the **DC_Block** and **DC_Feed** components (as appropriate) and place them in the circuit as follows:
   - Place the DC_Block component (represented by a capacitor) between the ports and the device under test.
   - Place the DC_Feed components (represented by an inductor, not shown in this design) between the pins of the device under test and any bias resistors.

S-parameter Simulation Overview

S-parameters are used to define the signal-wave response of an *n*-port electrical element at a given frequency. Detailed discussions of S-parameters can be found in standard textbooks on electrical circuit theory.

Note You may find it helpful to review the publication *S-Parameter Techniques for Faster, More Accurate Network Design* (AN 95–1), 5952–1130 (available through the Agilent Web site).

S-parameter simulation is a type of small-signal AC simulation. It is most commonly used to characterize a passive RF component and establish the small-signal characteristics of a device at a specific bias and temperature.

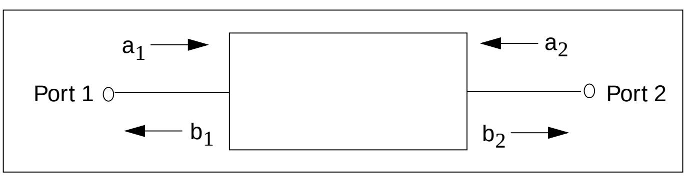
All nonlinear components are linearized and the linear circuit that results is analyzed as a multiport device. Each port is excited in sequence, a linear small-signal simulation is performed, and the response is measured at all ports in the circuit. That response is then converted into S-parameter data, which are in turn sent to the dataset. S-parameter simulation normally considers only the source frequency in a noise analysis. Use the Enable AC Frequency Conversion option if you also want to consider the frequency from a mixer's upper or lower sideband

---

Note By selecting the appropriate option under the *Parameters* tab, it is possible to convert S-parameter data to Y- and Z-parameters. The S-parameter results are retained.

S-parameter Definitions

The following is a representation of a signal wave in a two-port electrical-element.



where

a_1 is the wave into port 1
b_1 is the wave out of port 1
a_2 is the wave into port 2
b_2 is the wave out of port 2

The S-parameters for this conventional element are defined in standard microwave textbooks as follows:

$$b_1 = a_1 s_{11} + a_2 s_{12}$$

$$b_2 = a_1 s_{21} + a_2 s_{22}$$

where

s_{11} is the port-1 reflection coefficient: $s_{11} = b_1/a_1$; $a_2 = 0$
s_{22} is the port-2 reflection coefficient: $s_{22} = b_2/a_2$; $a_1 = 0$
s_{21} is the forward transmission coefficient: $s_{21} = b_2/a_1$; $a_2 = 0$
s_{12} is the reverse transmission coefficient: $s_{12} = b_1/a_2$; $a_1 = 0$

These equations can be solved for b_1 and a_1 in terms of a_2 and b_2 to yield the transmission (T) parameters as follows:

$$b_1 = a_2 t_{11} + b_2 t_{12}$$

$$a_1 = a_2 t_{21} + b_2 t_{22}$$

The T-parameters are related to the S-parameters as follows:

$$\begin{bmatrix} t_{11} & t_{12} \\ t_{21} & t_{22} \end{bmatrix} = \begin{bmatrix} s_{12} - s_{11} \ s_{22}/s_{21} & s_{11}/s_{21} \\ -s_{22}/s_{21} & 1/s_{21} \end{bmatrix}$$

S-parameters are defined with respect to a reference impedance that is typically 50 ohms. For 50-ohm S-parameters—with the 2-port element terminated with 50 ohms at each port—the s_{21} parameter represents the voltage gain of the element from port 1 to port 2.

Group Delay

Group delay is a useful measure of phase distortion in components such as amplifiers and filters. It measures the transit time, with respect to frequency, of a signal through the device under test.

The simulator calculates group delay by performing a finite difference of the phase response to obtain $d\Phi / d\omega$.

group_delay = *delay*(2,1) = –diff(unwrap(phaserad(S21),pi))/(2pi)

The simulator sets the frequency aperture to 0.01% of the current frequency. You can override this value by modifying the value in the *Group delay aperture* field, under the *Parameters* tab. This function is similar to that found on network analyzers, like the 8710. Refer also to these functions:

delay	dev_lin_phase	phaserad	unwrap
diff	phasedeg	ripple (and GpDelRip measurement equation)	volt_gain

Descriptions are in the *Expressions, Measurements, and Simulation Data Processing* manual.

Group delay results are considered with respect to the input and output ports only. Results of group delay calculations include *delay(2,1)* and *delay(1,2)*, which can be viewed in the Data Display. These are absolute group delay, in seconds. For additional results data, add the measurement equations *dev_lin_phase* and *GpDelRip* to the schematic. Calculations from these equations will also be available in the Data Display.

S-parameter Frequency Conversion

S-parameter simulation normally allows only one frequency to be considered in a noise analysis—that of the source. This can be a disadvantage in obtaining simulation results for circuits involving mixers, which are inherently frequency-translating devices involving multiple frequencies. (Refer to the chapter “Mixers” in the *Harmonic Balance Simulation* documentation.)

As an aid in the simulation of frequencies involving mixer subnetworks, the option *Enable AC Frequency Conversion* (under the *Parameters* tab) causes the simulator to consider not only the frequency of the source but also that of one of the mixer’s sidebands (which are defined by the user). Only the upper or lower sideband is considered, not both. The frequency-conversion results will appear in the dataset as for any nodes or probes placed to capture voltage or current data.

The S-parameter simulator uses the same conversion algorithm used by the AC Simulation component. For more information on this algorithm, refer to the topic “Enabling Frequency Conversion” in the *Linear AC and Noise Simulation* documentation.

Noise Analysis

During an S-parameter simulation, you can calculate these noise characteristics:

- Noise figure
- Noisy 2-port parameters

Each are discussed in the sections that follow.

Hint To aid in noise figure measurements, use the noise circle measurement component, *NsCircle* or the function *ns_circle*. Refer to *ns_circle* in the *Expressions, Measurements, and Simulation Data Processing* documentation.

Noise Figure

The parameter nf(k) in the dataset is the noise figure at output port k. You can view results using the Data Display. The noise figure for each port will appear in the variables list as nf(1), nf(2), and so on. nf displays noise figures for all ports.

When noise figure is calculated at a port, the other ports in the network are terminated in their respective impedances.

For a 2-port circuit, the noise figure is the signal-to-noise ratio at the input, divided by the signal-to-noise ratio at the output. It has default units of dB. For a circuit with more than two ports, the noise figure is the ratio of the total noise at the output port to the transmitted input noise. The total noise is the transmitted input noise plus the noise contributed by the network. The transmitted input noise represents the portion of the incident thermal noise (kTB, where k=Boltzmann's constant, 1.381...x10-23 J/K, T=290 K, B=1 Hz) which passes through the system.

Calculating Noise Figure

The common definition of noise factor is signal to noise at the input divided by the signal to noise at the output or

$$F = \frac{SNR_i}{SNR_o}$$

Another definition is provided by Van Der Ziel:

$$F = \frac{\overline{V_o^2}}{\text{contribution of the source to } \overline{V_o^2}}$$

This definition describes the way that noise is computed in Advanced Design System.

This noise of the network without the ports is computed and denoted by

$$\overline{V_N^2}$$

The port noise is computed separately, and Van Der Ziel's equation is written as:

$$F = \frac{KTB|S_{21}| + \dfrac{V_N^2}{R}}{KTB|S_{21}|^2}$$

If there are multiple ports, the noise figure at output port k is generalized as:

$$F = \frac{\displaystyle\sum_{j \neq k}^{N} KT_j \Delta F |S_{kj}|^2 + \dfrac{\overline{V_N^2}}{R_k}}{\displaystyle\sum_{j \neq k}^{N} KT_j \Delta F |S_{kj}|^2}$$

Note that the output port noise is never included in the summation of noise sources. This definition makes it possible to generalize the noise figure calculation to the case of n-port networks, and in the limiting case of N=2, the calculation agrees with the classic 2-port definition.

Noisy 2-port Parameters

NFmin, Rn and Sopt are the noisy two-port parameters. They describe the noise properties of a two-port and how the noise changes with respect to the source impedance. They describe circles of constant noise figure on the Smith chart.

NFmin is the minimum noise figure that the circuit can produce, when the source has the optimum reflection coefficient Sopt.

Rn is the noise resistance and controls how fast the noise increases as the source reflection coefficient changes from Sopt.

$$F = F_{min} + \frac{4r_n\left|\Gamma_{src} - \Gamma_{opt}\right|^2}{(1 - \left|\Gamma_{src}\right|^2)\left|1 + \Gamma_{opt}\right|^2}$$

where $\Gamma_{opt} = S_{opt}$, and Γ_{src} is the reflection coefficient of the source.

Reference: G Gonzalez, *Microwave Transistor Amplifiers*, Prentice-Hall, 1984, p 142.

S-Parameters from Various Input/Output Modes

S-parameter results can be simulated for designs using various combinations of input and output modes. These modes include differential, common, and single-ended. Advanced Design System offers the following component and examples supporting various techniques for working with S-parameters:

- *SP_Diff* is an instrument control component and is available in the *Simulation-Instruments* component library. See *Simulation > Instrument Controllers > SP_Diff (Differential-Mode S-Parameters)*.
- The example project *$HPEESOF_DIR/examples/RFIC/Wireline_Application_prj* contains the following designs and datasets that demonstrate simulations and mode-conversion equations. See the documentation for this project in *Examples > Wireline > Wireline Applications* then scroll to “Signal Integrity Simulations”.
  - Common Mode Impedance Simulation is demonstrated in the design *ckt_common_imp_ML_thick_metal.dsn*.
  - Differential Impedance Simulation is demonstrated in the design *ckt_diff_imp_ML_thick_metal.dsn*.
  - Differential and Common Mode S-Parameter Basics is demonstrated in *mixed_mode_basics.dds*.

S_Param Simulation Controller

The S_Param Simulation controller enables you to define the following aspects of the simulation:

- **Frequency**—Sweep type and associated characteristics.
- **Parameters**—Calculation of S-, Y-, or Z-parameters or group delay, as well as parameters related to status level for summary information and device operating-point levels.
- **Noise**—Parameters related to linear noise calculation (including port noise).
- **Output**—In ADS, selectively save simulation data to a dataset. For details, refer to the topic “Selectively Saving and Controlling Simulation Data” in the chapter “Simulation Basics” in the *Circuit Simulation* documentation.

  In RFDE, use *Outputs* > *Save Options* in the Analog Design Environment window.
- **Display (in ADS)**—Control the visibility of simulation parameters on the Schematic. For details, refer to the topic “Displaying Simulation Parameters on the Schematic” in the chapter “Simulation Basics” in the *Circuit Simulation* documentation.

Setting up a Sweep in ADS

Setting up the sweep portion of the simulation consists of two basic parts:

- Selecting the sweep type and setting the associated characteristics
- Optionally, specifying a sweep plan

To shorten simulation time in any parameter sweep, select a start point as close as possible to the convergence point and vary the parameter gradually. This yields better estimates for the next simulation, and achieves convergence more rapidly than if the parameter were changed abruptly. Simulator parameter names, as they appear in netlists and ADS schematics, are in parentheses.

Table 1-1. ADS S_Param Simulation Frequency Sweep Options

Frequency
Frequency

Table 1-1. ADS S_Param Simulation Frequency Sweep Options (continued)

Sweep Type—The sweep type and parameters (SweepVar="freq").	
Single point (Freq)	Enables simulation at a single frequency point. Specify the desired value in the Frequency field.
Linear	Enables sweeping a range of values based on a linear increment. Click Start/Stop to set start and stop values for the sweep, or Center/Span to set the center value and a span of the sweep.
Log	Enables sweeping a range of values based on a logarithmic increment. Click Start/Stop to set start and stop values for the sweep, or Center/Span to set the center value and a span of the sweep.
Start/Stop	Select the Start/Stop option to sweep based on start, stop, step-size or pts./decade, and number of points. Linear sweep uses Step-size; Log sweep uses Pts./decade. Start (Start)—the start point of a sweep Stop (Stop)—the stop point of a sweep Step-size (Step)—the increments at which the sweep is conducted Pts./decade (Dec)—number of points per decade Num. of pts. (Lin)—the number of points over which sweep is conducted
Center/Span	Select the Center/Span option to sweep based on center and span, step-size or pts./decade, and number of points. Linear sweep uses Step-size; Log sweep uses Pts./decade. Center (Center)—the center point of a sweep Span (Span)—the span of a sweep Step-size (Step)—the increments at which the sweep is conducted Pts./decade (Dec)—number of points per decade Num. of pts. (Lin)—the number of points over which sweep is conducted
Note: Changes to any of the Start, Stop, etc. fields causes the remaining fields to be recalculated automatically.	
Use sweep plan (SweepPlan)	Enables use of an existing sweep plan component (SweepPlan). Select this option and enter the name of the plan or select it from the drop-down list.

Setting up a Sweep in RFDE

Setting up the sweep portion of the simulation consists of two basic parts:

- Specifying the parameter type
- Specifying a sweep plan

To shorten simulation time in any parameter sweep, select a start point as close as possible to the convergence point and vary the parameter gradually. This yields better estimates for the next simulation, and achieves convergence more rapidly than if the parameter were changed abruptly.

Table 1-2. RFDE S-Parameters Simulation Parameter Sweep

Parameter Sweep	
Parameter Type	
Frequency	freq is automatically selected because it is a reserved variable name.
Design Variable	Click Select to choose a variable name from the Select Design Variable form. The list contains variables set up in the Editing Design Variables form. You can also type in a name in the Variable Name field. However, the variable must exist in the design for a successful simulation. Also enter a value for At Frequency.
Temperature	temp is automatically selected because it is a reserved variable name. Also enter a value for At Frequency.
Component Parameter	Click Select and choose a component from the schematic. In the Select Component Parameter form, select the parameter to be swept, then click OK. You can also type in the names in the Component Name and Parameter Name fields. However, the component and parameter must exist in the design for a successful simulation. Also enter a value for At Frequency.
Model Parameter	Type in the name of a defined model and the name of the model parameter to be swept. Also enter a value for At Frequency.
At Frequency	The frequency at which to sweep the variable.
Sweep Plan	
Choose one sweep range:	
Start-Stop	Sets the Start and Stop values of the sweep Start - The start point of the sweep Stop - The stop point of the sweep
Center-Span	Sets the Center value and a Span of the sweep. Center - The center point of a sweep Span - The span of a sweep

Table 1-2. RFDE S-Parameters Simulation Parameter Sweep

Choose one sweep type:	
Linear	Enables sweeping a range of values based on a linear increment. Set the increment with Step Size *or* Number of Steps. Use Additional Points to add specific values. Step Size - The increments at which the sweep is conducted Number of Steps - The number of points over which sweep is conducted
Logarithmic	Enables sweeping a range of values based on a logarithmic increment. Set the increment with Points Per Decade *or* Number of Steps. Use Additional Points to add specific values. Points Per Decade - The number of points per decade. Number of Steps - The number of points over which sweep is conducted
Points Only	Enables simulation at specific values for the parameter. Enter values in the Specific Points field with a space between each one.
Additional Points	When sweep type is Linear or Logarithmic, click this option to enter specific values to include in the sweep range. Enter values with a space between each one.

Defining Simulation Parameters

Defining the simulation parameters consists of the following basic parts:

- Enabling the frequency conversion (in ADS).
- Enabling the budget simulation (in ADS).
- Specifying the desired level of detail in the simulation status summary.
- Specifying the amount of device operating-point information to save.

Simulator parameter names, as they appear in netlists and ADS schematics, are in parentheses.

Table 1-3. S_Param Simulation Parameter Options

Parameters	
Calculate	
S-parameters (CalcS)	Causes S-parameters to be calculated.
Y-parameters (CalcY)	Converts the results of an S-parameter simulation to Y-parameters. The S-parameters are also output.
Z-parameters (CalcZ)	Converts the results of an S-parameter simulation to Z-parameters. The S-parameters are also output.
Group delay (CalcGroupDelay)	Causes a group delay simulation to be calculated from the S-parameter data. *Group delay aperture* is the frequency aperture (as a percentage of actual frequency) over which $d\Phi / d\omega$ is calculated.
Frequency Conversion (ADS only)	
Enable AC frequency conversion (FreqConversion)	Select this option to enable AC frequency conversion. For *S-parameter frequency conv. point* (FreqConversionPort) add a numeric value to enable S-parameter frequency.
Levels (in ADS) Annotation (in RFDE) Enables you to set the level of detail in the simulation status report.	

Table 1-3. S_Param Simulation Parameter Options (continued)

Status level (StatusLevel)	Prints information about the simulation in the Status/Summary part of the Message Window. - 0 reports little or no information, depending on the simulation engine. - 1 and 2 yield more detail. - Use 3 and 4 sparingly since they increase process size and simulation times considerably. The type of information printed may include the sum of the current errors at each circuit node, whether convergence is achieved, resource usage, and where the dataset is saved. The amount and type of information depends on the status level value and the type of simulation.
Device operating point level (DevOpPtLevel)—Enables you to save all the device operating-point information to the dataset. In ADS, if this simulation performs more than one S-Parameter analysis (from multiple S-Parameter controllers), the device operating point data for all S-Parameter analyses will be saved, not just the last one. Default setting is None.	
None (=None)	No information is saved.
Brief (=Brief)	Saves device currents, power, and some linearized device parameters.
Detailed (=Detailed)	Saves the operating point values which include the device’s currents, power, voltages, and linearized device parameters.

Defining Noise Parameters

Defining the noise parameters consists of the following basic parts:

- Enabling noise calculation.
- Specifying the nodes to use for noise parameter calculation.
- Specifying the noise contributors and the threshold for noise contribution.
- Optionally, specifying the bandwidth over which the noise simulation is performed.

Simulator parameter names, as they appear in netlists and ADS schematics, are in parentheses.

Table 1-4. S_Param Noise Parameter Options

Noise	
Calculate noise (CalcNoise) (in ADS) Noise (in RFDE)	Enables linear noise simulation.This option must be selected for the remaining noise options to be available.
Noise input port (NoiseInputPort) (ADS) Input Port Number (RFDE)	The number of the source port at which noise is injected. This is commonly the RF port. Although any valid port number can be used, the input port number is frequently defined as Num=1. In RFDE, use the Select Port button to select the input port from the schematic, or enter the port number directly in the input field.
Noise output port (NoiseOutputPort) (ADS) Output Port Number (RFDE)	The number of the Term component at which noise is retrieved. This is commonly the IF port. Although any valid port number can be used, the output port number is frequently defined as Num=2. In RFDE, use the Select Port button to select the output port from the schematic, or enter the port number directly in the input field.
Noise contributors—Use this area to sort the noise contributors list and to select a threshold below which noise contributors will not be reported. A list shows how each component contributes to noise at a specific node. The noise contributor data are always in units of V/sqrt(Hz) for noise voltages, and A/sqrt(Hz) for noise currents; they do not scale with noise bandwidth.	
Mode (SortNoise)	Provides options for sorting noise contributors by value or name.
Off (=Off)	Causes no individual noise contributors to be selected. The result is simply a value for total noise at the output.
Sort by value (=Sort by value)	Sorts individual noise contributors, from largest to smallest, that exceed a user-defined threshold (see below). The subcomponents of the nonlinear devices that generate noise (such as Rb, Rc, Re, Ib, and Ic in a BJT) are listed separately, as well as the total noise from the device.

Table 1-4. S_Param Noise Parameter Options (continued)

Sort by name (=Sort by name)	Causes individual noise contributors to be identified and sorts them alphabetically. The subcomponents of the nonlinear devices that generate noise (such as Rb, Rc, Re, Ib, and Ic in a BJT) are listed separately, as well as the total noise from the device
Sort by value with no device details (=Sort by value with no device details)	Sorts individual noise contributors, from largest to smallest, that exceed a user-defined threshold (see below). Unlike *Sort by value*, only the total noise from nonlinear devices is listed without any subcomponent details.
Sort by name with no device details (=Sort by name with no device details)	Causes individual noise contributors to be identified and sorts them alphabetically. Unlike *Sort by name*, only the total noise from nonlinear devices is listed without subcomponent details.
Dynamic range to display (NoiseThresh)	A threshold below the total noise, in dB, that determines what noise contributors are reported. All noise contributors less than this threshold will be reported. For example, assuming that the total noise voltage is 10 nV, a setting of 40 dB (a good typical value) ensures that all noise contributors up to 40 dB below 10 nV (that is, up to 0.1 nV) are reported. The default of 0 dB causes all noise contributors to be reported. If *Sort by name* is selected, accept the default.
Bandwidth (BandwidthForNoise)	The bandwidth over which the noise simulation is performed. 1 Hz is the recommended bandwidth for measurements of spectral noise power. The noise contributor data do not scale with noise bandwidth
Other	Check this box to enable access to hidden parameters. The text field is enabled to allow assigning values to the parameters. The format is `Other=HiddenParameter1=value HiddenParameter2=value`... Hidden parameters are used typically when troubleshooting convergence problems. (In ADS, display and set parameter directly on the schematic.)

Index

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

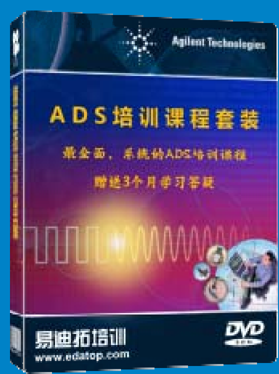
射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

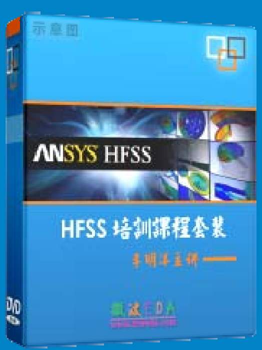
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...



课程网址： http://www.edatop.com/peixun/ads/13.html

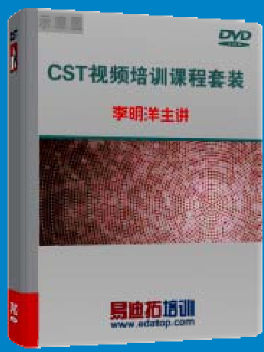


HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的HFSS培训教程套装，可以帮助您从零开始，全面深入学习HFSS的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装



该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com