Agilent Technologies

Advanced Design System 2011.01

Feburary 2011
AC Simulation

Acknowledgments

About Linear AC and Noise Simulation

The AC simulation component, in the Simulation-AC palette, performs a small-signal, linear AC analysis. As part of the analysis, the DC operating point is calculated and any nonlinear devices are linearized around that operating point. This analysis does not generate harmonics or exhibit compression. An AC simulation enables you to obtain small-signal transfer parameters, such as voltage gain, current gain, transimpedance, transadmittance, and linear noise.

Refer to the following topics for details on Linear AC and Noise simulation:

- *Performing an AC Simulation* (cktsimac) has the minimum setup requirements for an AC simulation.
- *Examples of AC Simulation* (cktsimac) describes in detail how to set up a basic AC simulation and how to calculate noise in ADS.
- *AC Simulation Description* (cktsimac) is a brief description of the AC simulator.
- *Linear Noise Simulation Description* (cktsimac) describes how noise is calculated as part of an AC simulation.
- *AC Simulation Parameters* (cktsimac) provides details about the parameters available in ADS for the AC simulation controller.

Performing an AC Simulation

An AC simulation is performed in the frequency domain. You can simulate a single frequency point, or across a frequency span in a linear or logarithmic sweep.

To perform an AC simulation, create your circuit, then add current probes and identify the nodes from which you want to collect data.

For a successful analysis, be sure to:

- Add the AC simulation component to the schematic. Double-click to edit it. Fill in the fields under the Frequency tab:

  > **Note**
  > If frequency conversion is *not* enabled, use the AC simulation component to specify the simulation frequencies-don't use the frequency parameters on the sources. If frequency conversion *is* enabled, use the sources to specify frequencies; do not use the AC simulation component's frequency entries.

  - Select the sweep type. For a single point, enter the frequency. For a linear or logarithmic sweep, elect to define the sweep with start/stop or center/span values.
- 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.
- You can enable frequency conversion, which is useful when analyzing circuits with standard (not user-defined) behavioral mixer models. For more information on this option, refer to *Enabling Frequency Conversion* (cktsimac).
- You can perform budget calculations as part of the simulation. For more information on budget analyses, see *Using Circuit Simulators for RF System Analysis* (cktsim).

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

Examples of AC Simulation

This section gives detailed setups to perform an AC simulation for:

- Analyzing a Power Amplifier
- Calculating Linear Noise
- Calculating Port Thermal Noise

The examples show how to simulate a power amplifier and display the amplifier output, how to calculate linear noise, and how to calculate port thermal noise.

Analyzing a Power Amplifier

The first figure below illustrates the setup for an AC simulation of a power amplifier .

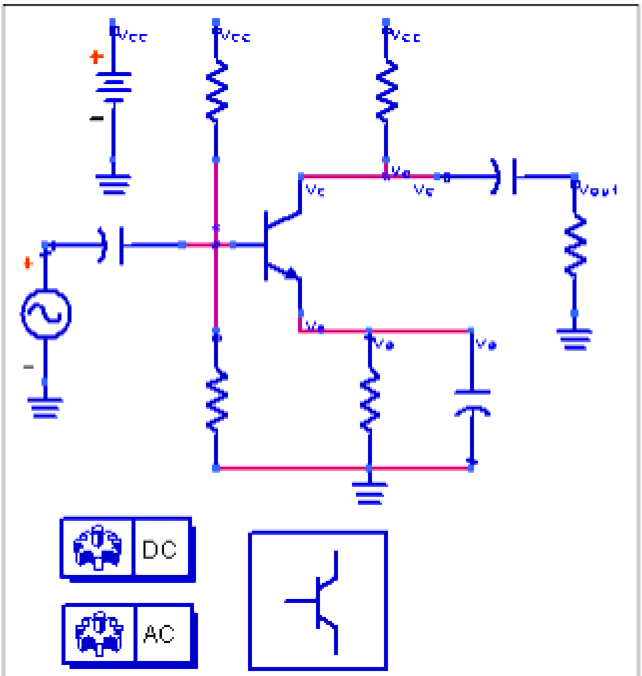
> **Note**
> This design, *AC1*, is in the *Examples* directory under *Tutorial/SimModels_wrk*. The results are in *AC1.dds*.

In this example, no variables are swept. The output load resistor has been labeled Vout, to represent the AC output voltage.

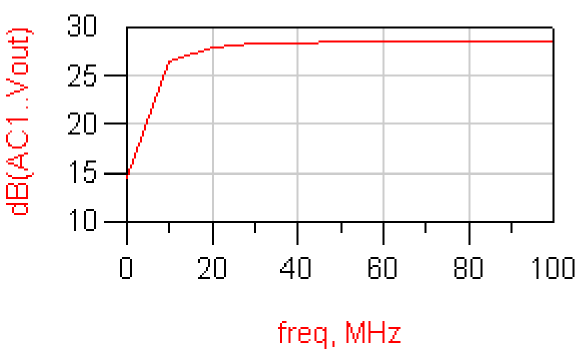
To perform a basic AC simulation:

1. From the **Sources-Freq Domain** palette, select **V_DC**. Place this component on the schematic and edit the component so that Vdc = **5 V**.
2. From the **Sources-Freq Domain** palette, select **V_1Tone**. Place this component on the schematic and edit the component so that Vac = **1V**. You can optionally use a V_AC component.
3. From the **Simulation-AC** palette, select **AC**. Place this simulation component on the schematic and edit it to select the **Frequency** tab. Ensure that *Start/Stop* is selected, then set the following values:
    - Sweep Type = **Linear**
    - Start = **1 kHz**
    - Stop = **100 MHz**
    - Step = **10 MHz**

    Figure: Example setup for a basic AC simulation



4. Click **OK** to accept changes and close the dialog box.
5. **Simulate**. When the simulation is finished, a Data Display window opens. Plot **Vout**. The following illustration shows a plot of AC output voltage (Vout), in dB, versus frequency:
   Figure: Plot of AC output voltage versus frequency



Calculating Linear Noise

You can simulate linear noise by setting the appropriate options via the *Noise* tab of the AC Simulation component.

To perform a linear noise analysis:

1. In the circuit to be simulated, ensure that you have named the nodes at which noise data are to be reported (for example, Input and Output).
2. Edit the AC Simulation component and select **Noise**, then select **Calculate Noise**.
3. In the *Edit* field, enter the names of nodes at which you want noise data to be reported.
4. Use the *Mode* popup menu to obtain the noise contributed by individual noise sources

and sort these contributions by name or value. To obtain just the total noise value at the output only, set Mode to **Off** .

5. Either accept the default for *Bandwidth* , or edit it as required. The default is often sufficient.
6. Enter a value in the *Dynamic range to display* field, in dB. The default is often sufficient if *Mode* is set to *On* .
7. Click **OK** to accept changes and close the dialog box.
8. Launch the simulation. To display and plot resulting data, see *Data Display* (data).

Calculating Port Thermal Noise

To simulate port thermal noise:

Proceed as in Calculating Linear Noise, with the following additional steps prior to launching the simulation:

- Ensure that you have placed input and output ports in the circuit.
- Select the *Include port noise* option.

For information on how noise is calculated, see *Linear Noise Simulation Description* (cktsimac).

AC Simulation Description

When an AC small-signal simulation is run, the system first computes the DC operating point of the circuit. Whenever a linear simulation such as a linear AC simulation requires a single-point DC bias simulation to be run first, it is referred to as a *bias-dependent* linear simulation. The most common example is the case of a linear amplifier that uses a biased transistor as the active element. The DC bias simulation is executed automatically and transparently (unless an error causes the DC simulation to fail to converge).

Following the DC bias simulation, the simulator linearizes all nonlinear devices about their bias points. A linearized model captures the small incremental changes of current due to small incremental changes of voltage. These are the derivatives of the transistor model equations, which are evaluated at the DC bias point. Nonlinear resistors and current sources are replaced by linear resistors whose values are set by the small signal conductance dI/dV. Current sources that depend on voltages other than the voltage across the source are replaced by linear dependent current sources dI_1/dV_2. Nonlinear capacitors are replaced by linear capacitors of value dQ/dV.

The resulting linear circuit is then simulated over the specified frequency range. Small-signal AC simulation is also performed before a harmonic-balance (spectral) simulation to generate an initial guess at the final solution.

Use the AC controller to:

- Perform a swept-frequency or swept-variable small-signal linear AC simulation.
- Obtain small-signal transfer parameters, such as voltage gain, current gain, transimpedance, transadmittance, and linear noise.

Simulation can be performed repeatedly while sweeping some parameter. If changing these parameters affects the DC operating point, the DC operating point and linearized circuit will be recomputed at each step.

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

Note
AC simulation is not intended to be used with Freq=0. The simulation will run, but would lead to unexpected results. If a DC response is desired, use DC controller.

Enabling Frequency Conversion

Traditional small-signal AC analysis is truly linear in the sense that frequency conversion effects do not occur. In RF system simulation, however, it is common to have frequency-translating mixer components that have approximately linear RF-to-IF conversion characteristics under small-signal RF drive. By enabling frequency conversion (also known as FCAC), you can perform system-level small-signal analyses on such systems. As is the case in standard AC analysis, a noise option is also available.

At the beginning of each FCAC simulation, a so-called *frequency map* is established. This map specifies the frequencies present at the various circuit nodes, and is based on the frequencies of the sources and the types of behavioral mixer components present in the network. Each node in the network can have only one frequency associated with it. Consequently, each behavioral mixer component can model frequency conversion to either the upper or the lower sideband, but not to both simultaneously.

Sources most often used for FCAC analysis include the V_1Tone, I_1Tone, and P_1Tone components. The frequency used by the source is given by the *Freq* parameter. If a multitone source is used, the frequency is specified by the *Freq[1]* parameter. When no frequency is explicitly specified, voltage and current sources default to the global value of the *freq* variable, while port sources simply become passive. Small-signal amplitudes used for FCAC analysis are given by *Vac*, *Iac*, and *Pac* parameters for voltage sources, current source, and ports, respectively.

Note
It is not possible to use FCAC analysis accurately with user-constructed circuit-level mixer blocks (such as diode mixers, Gilbert cell mixers, and the like).

Linear Noise Simulation Description

Linear noise simulation is an option available with the AC and S-parameter simulators. The frequency at which the noise is analyzed is the same as the AC simulation frequency. Noise voltages and currents are saved in the dataset with the keyword *Noise* included in the parameter name to identify the type of simulation.

The simulator performs an arbitrary-topology, multiport, network noise simulation. The following noise contributions are included in this simulation:

- Temperature-dependent thermal noise from lossy passive elements, including those specified by data files
- Temperature and bias-dependent noise from nonlinear devices
- Noise from linear active devices specified by 2-port data files that include noise parameters
- Noise from noise source elements

The noise simulation computes the noise generated by each element, and then determines how that noise affects the noise properties of the network. In most cases, the noise generated by circuit elements is calculated automatically. Lossy passive elements, for example, contribute noise according to their ability to deliver thermal noise power. The noise contributions from nonlinear devices are computed by models that include temperature and bias dependence; those models are similar to those used by SPICE. The computation of network-level noise properties from the component elements is performed by means of noise-correlation matrices. Most noise measurements are based on either *noise figure* or *noise parameter* calculations, which are defined for 2-port networks only. For networks with more than two ports, the noise figure can be measured between two user-specified ports using Input Port and Output Port; the other ports are treated as resistors for the noise simulation.

Note
The temperature of lossy passive elements is used to calculate their noise contributions. Since a lossy passive element at a physical temperature of 0 K does not generate any thermal noise, you may want to disable the noise contribution of any such element by setting its physical temperature to -273.15°C (0 K). Do not use this method for nonlinear devices such as transistors. Use the *Noise* parameter for resistors and nonlinear devices to enable (Noise = YES) or disable (Noise = NO) noise generation.

The program's nonlinear device models include one or more of the following noise effects:

- *Thermal noise* generated by the resistances that exist within the nonlinear device models. This noise is proportional to the device temperature and is independent of bias.
- *Channel noise* for JFET, MESFET, HEMT and similar devices. This noise may be due to thermal noise, high-field diffusion noise, or other effects. This noise is generally a function of device temperature and bias.
- *Shot noise* is caused by the quantized and random nature of current flow across junctions and is modeled for diodes and BJTs. This noise is proportional to the device bias current and is independent of temperature.
- *Flicker (1/f) noise* is modeled in most nonlinear devices.
- *Burst (or popcorn) noise* is another low-frequency, bias-dependent noise effect

modeled in bipolar transistors.

Noise Parameter Definitions

Noise parameters are used to define the noise electrical properties of an *n*-port electrical element at a given frequency. The noise parameters over a range of frequencies define the element's performance for all noise-power spectral density and tones that define an incident noise source.

Definitions of noise parameters can be found in standard textbooks covering electrical circuit theory. Noise parameters are used by the program to define the noise properties of any electrical element. The following discussion is for a 2-port element, but may be generalized for any *n*-port element.

A 2-port element noise-wave representation may use two noise waves at the element input. Otherwise, it may use one noise wave at the element input and one at the element output. A multiport-element noise-wave representation has one noise wave at each element port.

In the following noise discussions, the noise is assumed to be spot noise with a bandwidth of 1 Hz.

Note
The spot noise figure is the ratio of the output noise power per unit bandwidth to the portion of output noise power that is attributable to the thermal noise in the input termination per unit bandwidth. The noise temperature of the input termination is assumed to be 290 K.

The noise correlation matrix, [N], is defined as follows:

$$[N] = \begin{bmatrix} N_{11} & N_{12} \\ N_{21} & N_{22} \end{bmatrix} = \begin{bmatrix} \langle bn\ \ bn^* \rangle & \langle bn\ \ an^* \rangle \\ \langle bn^*\ \ an \rangle & \langle an\ \ an^* \rangle \end{bmatrix}$$

where * represents the complex conjugate.

Noise Entries in a Dataset

vout.noise	Total noise voltage at node Vout
vnc	Noise contributors (noise voltage contribution from each noise generator in the circuit)
name	Instance name of noise contributor

These entries can be viewed in the data display. For example, in a simple circuit with two noise generators, resistors R1 and R2, plotting *name* and *vnc* on a list, generates the following table:

name	vnc
_total	641.6 pV
R1	453.7 pV
R2	453.7 pV

These results show that R1 and R2 are each contributing an equal amount of noise to the total. Notice that the sum of the individual contributors does not add up to the value for _*total*. The sum of the squares of the individual noise contributors is equal to the total noise voltage squared.

Note
vout.noise reports that same noise voltage as the value of *vnc* associated with the name _*total*.

AC Simulation Parameters

ADS provides access to AC simulation parameters enabling you to define aspects of the simulation listed in the following table:

Tab Name	Description	For details, see...
Frequency	Sweep type and associated characteristics.	Setting Frequency Sweep
Noise	Parameters related to linear noise calculation (including port noise).	Defining Noise Parameters
Parameters	Provides options to set the following: • Frequency conversion • Performing budget simulation • Status levels for summary information • Device operating point information level	Defining Simulation Parameters
Output	Selectively save simulation data to a dataset.	*Selectively Saving and Controlling Simulation Data* (cktsim).
Display	Control the visibility of simulation parameters on the schematic.	*Displaying Simulation Parameters on the Schematic* (cktsim).
	Additional parameters that you may find useful.	Additional Parameters

Setting Frequency Sweep

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. The following table describes the parameter details. Names listed in the *Parameter Name* column are used in netlists and on schematics.

Table: AC Simulation Frequency Parameters

Setup Dialog Name	Parameter Name	Description
Frequency		
Sweep Type-The sweep type and parameters.		
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 Start, Stop, Step-size, Pts./decade, Num. of pts.	Start Stop Step Dec Lin	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-the start point of a sweep - Stop-the stop point of a sweep - Step-size-the increments at which the sweep is conducted - Pts./decade-number of points per decade - Num. of pts.-the number of points over which sweep is conducted
Center/Span Center, Span, Step-size, Pts./decade. Num. of pts.	Center Span Step Dec Lin	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-the center point of a sweep - Span-the span of a sweep - Step-size-the increments at which the sweep is conducted - Pts./decade-number of points per decade - Num. of pts.-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.

Defining Noise Parameters

Defining the noise parameters consists of the following basic parts:

- Enable 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.

The following table describes the parameter details. Names listed in the *Parameter Name* column are used in netlists and on schematics.

Table: AC Simulation Noise Parameters

Setup Dialog Name		Parameter Name	Description
Calculate noise		CalcNoise	This check box enables linear noise simulation. Voltages are reported for nodes, currents are reported for branches. See ns_circle
Nodes for noise parameter calculation-Use this area to select nodes at which you want linear noise data to be reported. Noise voltages and currents are reported in rms units.			
	Edit		Enables selection of nodes at which you want noise data reported.
	Select	NoiseNode	Contains the list of named node(s) at which noise parameters are to be reported. First use the Edit field to add named nodes to this window. Add - enables you to add a named node. Cut - enables you to delete a named node. Paste - enables you to take an item that has been cut and place it in a different order.
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: Dynamic range to display). 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 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: Dynamic range to display). 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. This parameter is used only with *Sort by value* and *Sort by value with no device details*.
Include port noise in node noise voltages		IncludePortNoise	Causes the simulator to model noise at input and output ports.
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.

Defining Simulation Parameters

Defining the simulation parameters consists of the following basic parts:

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

The following table describes the parameter details. Names listed in the *Parameter Name* column are used in netlists and on schematics.

Table: AC Simulation Parameters

Setup Dialog Name	Parameter Name	Description
Frequency Conversion		
Enable AC frequency conversion	FreqConversion	Causes a frequency-converting AC analysis to be performed.
Budget		
Perform Budget simulation	OutputBudgetIV	Enables Budget simulation, which reports current and voltage data at the pins of devices following a simulation. Current into the nth terminal of a device is identified as ...device_name.tn.i. Voltage at the nth terminal of a device is identified as ...device_name.tn.v.
Levels		Enables you to set the level of detail in the simulation status report.
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	Options to save device operating-point information for most active devices and some linear devices in the circuit to the dataset. In ADS, if this simulation performs more than one AC analysis (from multiple AC controllers), the device operating point data for all AC 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.

Additional Parameters

The following table includes additional parameter(s) that you may find useful.

Table: Additional Parameters

Setup Dialog Name	Parameter Name	Description

Other		Use *Other* to enable access to hidden parameters, and assign values to them. The format is:
		`Other=HiddenParameter1=value1`
		`HiddenParameter2=value2`
		`...`
		Hidden parameters typically are used when troubleshooting convergence problems. To set this parameter in ADS, select the parameter name on the Display tab, then enter the value directly on the schematic.