



ADS 2008
January 2008
Large-Signal S-Parameter Simulation

Advanced Design System 2008

© Agilent Technologies, Inc. 2000-2008

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

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

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. Java™ is a U.S. trademark of Sun Microsystems, Inc. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC.

Errata The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Warranty The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

Technology Licenses The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license. Portions of this product include the SystemC software licensed under Open Source terms, which are available for download at <http://systemc.org/>. This software is redistributed by Agilent. The Contributors of the SystemC software provide this software "as is" and offer no warranty of any kind, express or implied, including without limitation warranties or conditions or title and non-infringement, and implied warranties or conditions merchantability and fitness for a particular purpose. Contributors shall not be liable for any damages of any kind including without limitation direct, indirect, special, incidental and consequential damages, such as lost profits. Any provisions that differ from this disclaimer are offered by Agilent only.

Restricted Rights Legend If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Contents

- About Large-Signal S-Parameter Simulation
- Performing a Large-Signal S-Parameter Simulation
- Example of Large-Signal S-Parameter Simulation
 - Large-signal S-parameter simulation example in ADS
- LSSP Simulation Description
 - LSSP Simulation Process
 - Comparing LSSP and S-Parameter Simulations
- LSSP Simulation Parameters
 - Setting Up Ports
 - Defining Simulation Parameters

About Large-Signal S-Parameter Simulation

The LSSP Simulation controller in the Simulation-LSSP palette (ADS) computes S-parameters for nonlinear circuits such as those that employ power amplifiers and mixers. In the latter case, S-parameters can be computed across frequencies, that is, from the RF input to the IF output. LSSP simulation is based on the harmonic balance simulation and uses harmonic balance techniques.

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

- [Performing a Large-Signal S-Parameter Simulation](#) shows the minimum setup requirements for a large-signal S-parameter simulation.
- [Example of Large-Signal S-Parameter Simulation](#) is a detailed setup for calculating large-signal S-parameters, using a BJT as the example.
For an example of an LSSP analysis of a mixer circuit, refer to the design LSSP2.dsn, and its associated dataset, LSSP2.dds. It is in the same project folder as the BJT example. This example illustrates how to use a term at the output port, and uses a voltage source at the LO.
- [LSSP Simulation Description](#) is a brief explanation of the LSSP simulator and how it compares to the S-parameter simulator.
- [LSSP Simulation Parameters](#) provides details about the parameters available in the LSSP Simulation controller in ADS.



Note

You must have the LSSP simulator license to run the simulation. You can build the examples described in these topics without the license, but you will not be able to simulate them.

Performing a Large-Signal 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:

- Apply ports to all inputs and outputs. Use a P_1Tone or P_nTone power source to drive a port. Terminate other ports using port-impedance terminations (Term). Verify impedance.

Note
The power level at a passive port (Term) will be calculated by turning on power sources and measuring the power at the port; this value will be used to drive the port.

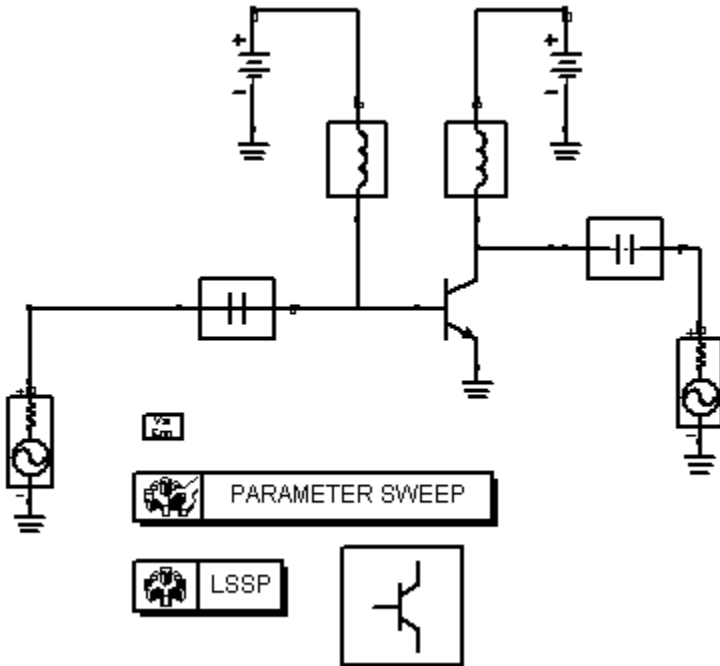
- Check the Num field for each port. The S-parameter port numbers are derived from these fields. For a 2-port circuit, you want the input labeled as Num = 1 and the output as Num = 2.
- In circuits with mixers, use a voltage source for the LO, not a power source. This prevents the LO input from being recognized as a port and consequently having the S-parameters calculated with respect to it.
- Add the LSSP control element to the schematic, then double-click to edit its parameters in the fields under the Freq and Ports tabs:
 - For Freq, set the fundamental frequency and order.
 - For Ports, set the port frequency for each port. S-parameters will be measured at this frequency. Port frequency can be the input frequency or a harmonic.
 - For choosing a matrix solver, the Auto Select mode is the default and recommended choice. For instructions about using this option, see [Selecting a Solver](#) in the [Harmonic Balance Simulation](#) documentation.
- You can use previous simulation solutions to speed up the simulation process. For more information, see [Reusing Simulation Solutions](#) in the [Harmonic Balance Simulation](#) documentation.

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

Example of Large-Signal S-Parameter Simulation

The following figure illustrates the setup for a large-signal S-parameter (LSSP) simulation of a BJT. Power sources drive all ports in this example.

Note
This design, LSSP_test.dsn, is in the Examples directory under Tutorial/LSSP_test_prj. The results are in LSSP_test.dds.



Large-signal S-parameter simulation example in ADS

1. From the Simulation-LSSP palette, select a P_1Tone component and place it at the input of the circuit. Edit the component and set the following values:
 - Num = 1
 - P = dbmtow((10),0)
 - Freq = LSSP_freq
2. Select another P_1Tone power source and place it at the output of the circuit. The values are the same as for the input source, except here Num = 2.

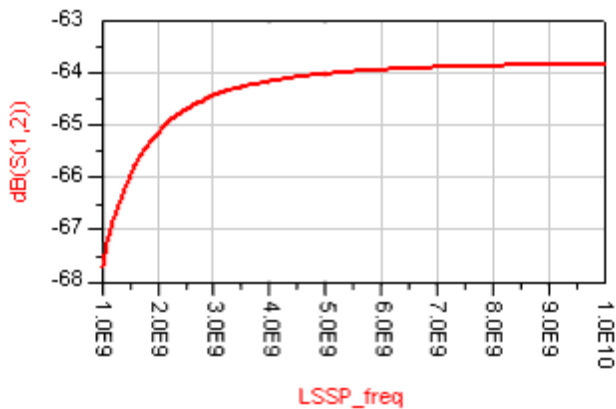
Note
 The values in the Num parameter on the sources and terminations should reflect the placement of the ports in the circuit, so that the S-parameter data is meaningful. The number of the input (source) should be set to Num=1, and that of the output (load) to Num=2.

3. Select and place an LSSP simulation component on the schematic, edit it, and select the Freq tab to set the following parameters:
 - Frequency = LSSP_freq
 - Order[1] = 3
 Click Add. Make sure that 1 LSSP_freq 3 is the only line that appears in the list of fundamental frequencies. (If LSSP_freq appears as the second fundamental in the frequency list, select the line above it and click Cut.)
4. Select the Sweep tab. Ensure that Start/Stop is selected and Sweep Type is Linear, then set the following values:
 - Start = 0
 - Stop = 100
 - Step = 1
5. Select the Ports tab. In the Frequency field, at the right of the dialog box, enter LSSP_freq and click Add. This establishes the frequency at port 1, where the large-signal S-parameter will be measured. It does not have to be

the same value as the fundamental frequency, it can be a harmonic.

For example, a port may have harmonics present at 0 Hz, 1 MHz, 99 MHz, 100 MHz, and 101 MHz. You can then specify your interest in the 99 MHz component by entering 99 MHz here. On the schematic, this appears as LSSP_FreqAtPort[n]. In this example, it would appear as LSSP_FreqAtPort[2]=99 MHz.

6. To set the frequency for port 2, click Add again. You should see two entries in the Port Frequency list box, each set to LSSP_freq.
7. Click OK to accept changes and close the dialog box.
8. From the Data Items palette, select VarEqn. Place and edit the component to define the variable LSSP_freq and set its value. Select the default equation ($X=1.0$). In the Variable Value field at the right, enter 1.0 GHz.
9. Click OK to accept changes and close the dialog box.
10. Since the fundamental is set to a single frequency, you can use a ParamSweep component to sweep a frequency range. Return to Simulation-LSSP, select and place a ParamSweep component, and edit it. Select the Sweep tab and set the following values:
 - Parameter to sweep = LSSP_freq
 - Sweep Type = Linear
 - Enable Start/Stop
 - Start = 1 GHz
 - Stop = 10 GHz
 - Step-size = 0.1 GHz
11. Click Simulations and set Simulation 1 to HB1. Click OK.
12. Launch the simulation, and when it is finished, display the results. LSSP data items may be identified with an HB prefix. The following plot displays $S(1,2)$.



LSSP Simulation Description

Unlike small-signal S-parameters, which are based on a small-signal simulation of a linearized circuit, large-signal S-parameters are based on a harmonic balance simulation of the full nonlinear circuit. Because harmonic balance is a large-signal simulation technique, its solution includes nonlinear effects such as compression. This means that the large-signal S-parameters can change as power levels are varied. For this reason, large-signal S-parameters are also called power-dependent S-parameters.

Like small-signal S-parameters, large-signal S-parameters are defined as the ratio of reflected and incident waves:

$$S_{ij} = \frac{B_i}{A_j}$$

The incident and reflected waves are defined as:

$$A_j = \frac{V_j + Z_{0j}I_j}{2\sqrt{R_{0j}}}, \quad B_i = \frac{V_i - Z_{0i}^*I_i}{2\sqrt{R_{0i}}}$$

where

V_i and V_j are the Fourier coefficients, at the fundamental frequency, of the voltages at ports i and j ,
 I_i and I_j are the Fourier coefficients, at the fundamental frequency, of the currents at ports i and j ,
 Z_{0i} and Z_{0j} are the reference impedances at ports i and j , and
 R_{0i} and R_{0j} are the real parts of Z_{0i} and Z_{0j} .

This definition is a generalization of the small-signal S-parameter definition in that V and I are Fourier coefficients rather than phasors. For a linear circuit, this definition simplifies to the small-signal definition.

LSSP Simulation Process

The simulator performs the following operations to calculate the large-signal S-parameters for a two-port:

- Terminates port 2 with the complex conjugate of its reference impedance. Applies a signal with the user-specified power level P_1 at port 1, using a source whose impedance equals the complex conjugate of that port's reference impedance. Using harmonic balance, calculates the currents and voltages at ports 1 and 2. Uses this information to calculate S_{11} and S_{21} .
- Terminates port 1 with the complex conjugate of its reference impedance. Applies a signal of power $P_2 = |S_{21}|^2 P_1$ at port 2 using a source whose impedance equals the complex conjugate of the reference impedance of port 2. Using harmonic balance, calculates the currents and voltages at ports 1 and 2. Uses this information to calculate S_{12} and S_{22} .

Comparing LSSP and S-Parameter Simulations

S-parameter simulations are performed on linear circuits. LSSP simulations can be performed on nonlinear circuits and

thus include nonlinear effects such as gain compression and variations in power levels.

Both LSSP and S-parameter simulations generate PortZ[] and S[] fields in the associated dataset. LSSP generates the additional field PortPower[], which contains the power, in dBm, seen at each port for the respective LSSP port frequencies.

To compare LSSP and S-parameter simulations, refer to LSSP1.dsn and SP1.dsn. They are in the ADS examples directory under Tutorial/SimModel_prj. The data displays are LSSP1.dds and SP1.dds.

For a review of the S-parameter simulator, see [S-Parameter Simulation Description](#).

LSSP Simulation Parameters

The parameters for LSSP Simulation are identical to those for Harmonic Balance, with these exceptions:

- The simulation component has a Ports tab (see [Setting Up Ports](#)).
- The Params tab contains additional parameters including selected options for Initial Guess (see [Defining Simulation Parameters](#)).
- The parameters for Small-signal mode, Nonlinear noise, and Oscillator are not available.

For details about the parameters not described here, see the section [HB Simulation Parameters](#).

Setting Up Ports

Set up the ports portion of the simulation using the information shown in the following table. Names listed in the Parameter Name column are used in netlists and schematics.

LSSP Simulation Ports Parameters

Setup Dialog Name		Parameter Name	Description
Port Frequency		LSSP_FreqAtPort[n]	Ports must be placed and defined. Set the port number of the input source to 1, and the port number of the output Term (termination) component to 2.
Select			Contains the list of fundamental frequencies. Use the Edit field to add fundamental frequencies to this window. - Add enables you to add an item. - Cut enables you to delete an item. - Paste enables you to take an item that has been cut and place it in a different order.
Edit			Edit the Frequency field, then use the buttons to Add the frequency to the list displayed under Select.
	Frequency		The frequency of the fundamental(s). Change by typing over the entry in the field. Select the units (None, Hz, kHz, MHz, GHz) from the drop-down list.

Defining Simulation Parameters

Defining the LSSP simulation parameters in ADS consists of these basic parts:

- Specifying the amount of device operating-point information to save
- Setting FFT oversampling ratio
- Setting the Initial Guess parameters

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

Advanced Design System 2008

LSSP Simulation Parameter Options

Setup Dialog Name	Parameter Name	Description	
Device operating point level	DevOpPtLevel	Enables you to save all the device operating-point information to the dataset. 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.
FFT			
	Fundamental Oversample	FundOversample	Sets the FFT oversampling ratio. Higher levels increase the accuracy of the solution by reducing the FFT aliasing error and improving convergence. Memory and speed are affected less when the direct harmonic balance method is used than when the Krylov option is used.
	More...	Oversample[n]	Displays a small dialog box. To increase simulation accuracy, enter in the field an integer representing a ratio by which the simulator will oversample each fundamental.
Initial Guess			
	Use Initial Guess	UseInFile	Check this box to enter a file name for a solution to be used as initial guesses. This file is typically generated from a previous simulation by enabling Write Final Solution. If no

			<p>initial guess file name is supplied, a default name (using DC solution) is generated internally, using the design name and appending the suffix .hbs. A suffix is neither required nor added to any user-supplied file name. For example, if you have saved the Harmonic Balance solution from a previous simulation, you can later do a nonlinear noise simulation and use this saved solution as the initial guess, removing the time required to recompute the nonlinear Harmonic Balance solution. Or you could quickly get to the initial Harmonic Balance solution, then sweep a parameter to see the changes. In this latter case, you will probably either want to disable the Write Final Solution option or use a different file name for the final solution to avoid overwriting the initial guess solution. See Write Final Solution below (parameter: UseOutFile).</p> <p>The Annotate value specified in the DC Solutions tab in the Options block is also used to control the amount of annotation generated when there are topology changes detected during the reading of the initial guess file. Since HB simulations also utilize the DC solution, to get optimum speed-up, both the DC solution and the HB solution should be saved and re-used as initial</p>
--	--	--	---

			<p>guesses.† The initial guess file does not need to contain all the HB frequencies. For example, one could do a one-tone simulation with just a very nonlinear LO, save that solution away and then use it as an initial guess in a two tone simulation. The exact frequencies do not have to match between the present analysis and the initial guess solution. However, the fundamental indexes should match. For example, a solution saved from a two tone analysis with Freq[1]=1GHz and Freq[2]=1kHz would not be a good match for a simulation with Freq[1]=1kHz and Freq[2]=1 GHz. If the simulator cannot converge with the supplied initial guess, it then attempts to a global node-setting by connecting every node through a small resistor to an equivalent source. It then attempts to sweep this resistor value to a very large value and eventually tries to remove it.</p>
	File	InFile	Specify a filename to save results.
	Regenerate Initial Guess for ParamSweep (Restart)		
Final Solution			
	Write Final Solution	UseOutFile	Check this box to save your final HB solution to the output file. If a filename is not supplied, a file name is internally generate using

			<p>the design name, followed by an .hbs suffix. If a file name is supplied, the suffix is neither appended nor required. If this box is checked, then the last HB solution is put out to the specified file. If this is the same file as that used for the Initial Guess, this file is updated with the latest solution.</p> <p>Transient simulations can also be programmed to generate a harmonic balance solution that can then be used as an initial guess for an HB simulation.</p> <p>‡</p>
	File	OutFile	Specify a filename to save results.
Harmonic Balance Assisted Harmonic Balance		HBAHB_Enable	Set the HBAHB mode to Auto, On, or Off. Default is Auto and is recommended.

† See [DC Simulation](#).

‡ See [Harmonic Balance Simulation](#).