Agilent Technologies

SPICE Model Generator

August 2005

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

Use, duplication or disclosure by the U. S. Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 for DoD agencies, and subparagraphs (c) (1) and (c) (2) of the Commercial Computer Software Restricted Rights clause at FAR 52.227-19 for other agencies.

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

Acknowledgments

Contents

Chapter 1: SPICE Model Generator

The SPICE Model Generator takes S-parameter data as input and produces an approximate equivalent model in SPICE format. The S-parameter data may be anywhere from 1 to 99 port S-parameter data. This data can originate from an Advanced Design System (ADS) S-parameter simulation, a High Frequency Structure Simulator (HFSS) or Momentum electromagnetic simulation, or a network analyzer measurement. The SPICE Model Generator processes the S-parameter data then generates a subcircuit containing HSpice or MDIF elements.

Available SPICE models include:

- Lumped PI-Network Model
- N-Section Ladder (rise time)
- N-Section Ladder (sections)
- Lossless Transmission Line(s)
- Rational Polynomial (Hspice)
- Rational Polynomial (MDIF)

The first four models are low-frequency extractions from S-parameter data. The fifth and sixth models, the rational polynomial models, are derived from curve-fitting-to-frequency data. For the first four models, the frequency data points in the S-parameter data file must be noise free. When the S-parameter data is taken from a network analyzer, the resulting low-frequency noise may necessitate the use of one of the rational polynomial models.

The SPICE model generator can read S-parameter files from an ADS Dataset, or Touchstone format. You can export ADS S-parameter data to one of these formats using the *Data File Tool*. For detailed information on using the data file tool, refer to Chapter 4, Working with Data Files, in *Using Circuit Simulators.*

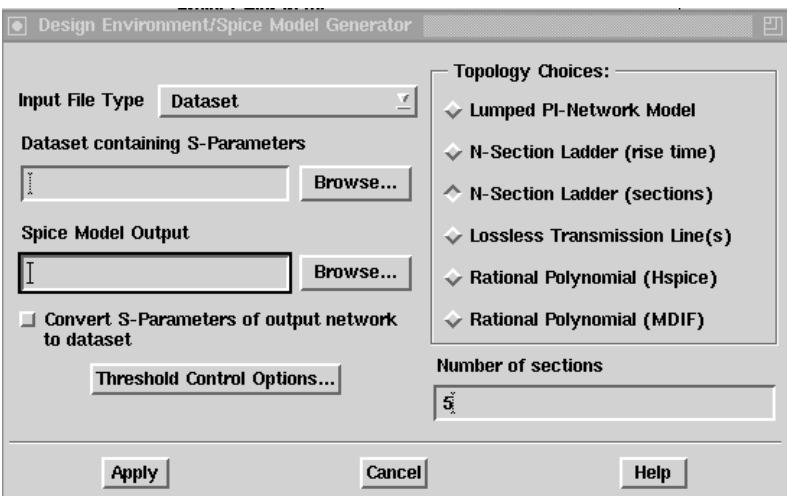
The SPICE model solution is stored as a Spice netlist—an ASCII file with a *.cir* suffix. Note that if you use any extension other than *.cir* the **Apply** button will not operate. If you leave the extension blank, SPICE model generator will automatically append the extension to the file.

Note As an alternative to using the SPICE Model Generator, you can define your own topology. To do this, create an equivalent circuit model and use the optimization function in ADS to optimize the parameter values of the equivalent circuit to match the S-parameter data file. For details on performance optimization, refer to the ADS *Tuning, Optimization and Statistical Design* documentation.

The information contained in the remainder of this document describes the dialog boxes for the SPICE Model Generator.

Design Environment/Spice Model Generator

The Design Environment/Spice Model Generator dialog box enables you to translate S-parameters into a SPICE netlist.



Input File Type

Specifies the input file type. Choices are Dataset or Touchstone File.

Dataset containing S-Parameters

Specifies the input file containing the S-parameters. You can enter this manually or use the **Browse** button to specify the directory and file name.

Spice Model Output

Specifies the name of the file that the SPICE model will be written to. You can enter this manually or use the **Browse** button to specify the directory and file name.

Convert output S-Parameters to dataset

Generates an ADS dataset from the existing input file.

Threshold Control Options

Opens a dialog for specifying threshold options. See “Threshold Control Options” on page 1-5.

Topology Choices

You can select a topology from the list. See “Topology Choices” on page 1-4.

Risetime

Enter the risetime in nanoseconds for N-Section Ladder (rise time) topology.

Number of sections

Enter the number of sections for N-Section Ladder (sections) topology.

Number of poles

Enter the number of poles for Rational Polynomial topologies.

Related Topics

"Topology Choices" on page 1-4.

"Threshold Control Options" on page 1-5.

Topology Choices

- Lumped PI-Network Model—A model consisting of resistance, inductance, capacitance, conductance, and mutual inductance elements. A PI- Network is defined as a capacitor in parallel with a resistor at each port or an inductor in series with a resistor between two ports.
- N-Section Ladder (rise time)—Ladder network topology, where N is the number of sections. The model consists of resistance, inductance, capacitance, conductance, and mutual inductance elements. For an N-Section Ladder network the formula for translating *rise_time_in_ns* into N sections is:

  Nsections = 15 x (delay on the line (in ns))/riseTime (in ns)
- N-Section Ladder (sections)—Ladder network topology, where N is the number of sections. The model consists of resistance, inductance, capacitance, conductance, and mutual inductance elements. For N-Section networks, the more sections you have the more accurate your model becomes.

Note N-Section Ladder networks are only valid for Transmission Line type circuits, whereas a PI-Network model can be used for all circuits.

- Lossless Transmission Line(s)—A model with true distributed values (Z0, T_d). It is useful for printed circuit board traces.
- Rational Polynomial (Hspice)—A behavioral model (mathematical expression) supported by Hspice. The model is a single polynomial element that can represent complex, arbitrary network structures.
- Rational Polynomial (MDIF)—A behavioral model (mathematical expression) supported by Measurement Data Interchange Format. The model is a single polynomial element that can represent complex, arbitrary network structures.

Related Topics

"Design Environment/Spice Model Generator" on page 1-2.

"Threshold Control Options" on page 1-5.

Threshold Control Options

Sets the thresholds for the model.

Capacitance

Sets the minimum capacitance, in picofarads.

Inductance

Sets the minimum inductance, in nanohenries.

Resistance

Sets the minimum resistance, in ohms.

Conductance

Sets the minimum conductance, in mhos.

Max Fitting Error

Sets the degree of error allowed in the model, in percent.

Npole - Nzero

Specifies the difference between the number of poles and the number of zeros in a rational polynomial model.

Number of Iterations

Specifies the number of iterations performed for a rational polynomial model.

Fitting Parameters

Specifies the type of parameters to fit into a rational polynomial model. Choices are Y or S.

Reset

Returns to default settings.

Cancel

Abort the Threshold Control Options setup operation.

Related Topics

"Design Environment/Spice Model Generator" on page 1-2.

"Topology Choices" on page 1-4.

Open File

Establishes the file to be opened.

Filter

Displays the full file path and file extension of the filter. Only files that are in this path with the specified extension are listed in the **Files** field. If you change the filter, click the **Filter** button to update the display.

Dataset S-parameter input files use the extension *.ds*.

If the number of ports for Touchstone S-parameter input files is < 10, use the extension *.sNp*, where *N* is the number of ports (e.g., a file containing the S-parameters for 4-port circuit would have an extention of *.s4p*). When the number of ports is 10 or more use the extention *.sNN* (where *NN* is the number of ports).

The default value in the file filter for output file types is the *.cir* extension.

Note If you use any extention other than *.cir* the **Apply** button will not operate. If you leave the extention blank, SPICE model generator will automatically append the extention to the file.

Directories

Displays directories for navigating to different file paths. Double-click the directory name. Double-click /.. to move up a level.

Files

Displays files that match filter specifications. Double-click the file name to select the file.

Selection

Displays the full file path of the selected directory. You type a full file path and filename in this field and the file will be selected.

Cancel

Abort the Open File dialog.

Index