Agilent Technologies

Advanced Design System 2002

SPICE Model Generator

February 2002

Notice

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

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, Touchstone or Citifile format. You can export ADS S-parameter data to one of these formats using the Instrument Server.

The SPICE model solution is stored as a Spice netlist—an ASCII file with a *.cir* suffix.

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 *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, Touchstone File or Citifile.

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

- Lumped PI-Network Model—A model consisting of resistance, inductance, capacitance, conductance, and mutual inductance elements.

- N-Section Ladder (rise time)—Ladder network topology, where N is the risetime. The model consists of resistance, inductance, capacitance, conductance, and mutual inductance elements.

- 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.

- 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

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 1-3.

"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*.

Touchstone File S-parameter input files use the extension .s*p, where * is the number of ports. For example, a file containing the S-parameters of a 4-port circuit should end in .s4p.

Citifile S-parameter input files use the extension *.citi*.

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

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

S
SPICE Model Generator, 1-1
    help for, 1-3
    threshold control options, 1-5
    topologies, 1-4