Broadband SPICE Model Generator

Agilent Technologies

Notices

Manual Part Number

00000-00000

Edition

January 2006

Printed in USA

Agilent Technologies, Inc.
395 Page Mill Road
Palo Alto, CA 94303 USA

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.

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.

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.

Safety Notices

CAUTION

A **CAUTION** notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a **CAUTION** notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Contents

1
Broadband SPICE Model Generator

This chapter introduces the Broadband SPICE Model Generator, discusses its benefits and provides an overview of the Broadband SPICE Model Generator use model. It also introduces the Broadband Spice UI, including the standard, advanced input and advanced output option tabs.

Broadband SPICE Model Generator Overview

The Broadband SPICE Model Generator tool converts frequency domain network parameter data of passive components (obtained by measurements and/or simulations) into equivalent circuit models for direct use with commercially available circuit simulators.

The Broadband SPICE Model Generation tool supports input model data available in a number of different file formats. This includes measured data, as well as, data generated by EM and frequency-domain simulators for ease of use. It also supports output model data in several different file formats, so that the data can be easily used with a number of different circuit simulators (e.g., transient, frequency domain, envelope and harmonic balance).

Supported Operating Systems

Broadband SPICE Model Generator is supported on all operating systems supported by ADS. See the *ADS Installation Manual* for more details.

Accessing the Documentation

To access the documentation for Broadband SPICE Model Generator, from any ADS window select **Help** > **Topics and Index** > **Model Development** > **Broadband SPICE Model Generator**, or from the Broadband SPICE Model Generator main menu select **Help** > **Topics and Index.**

Broadband SPICE Model Generator Use Model

The following diagram outlines the process flow for the Broadband SPICE Model Generator engine:

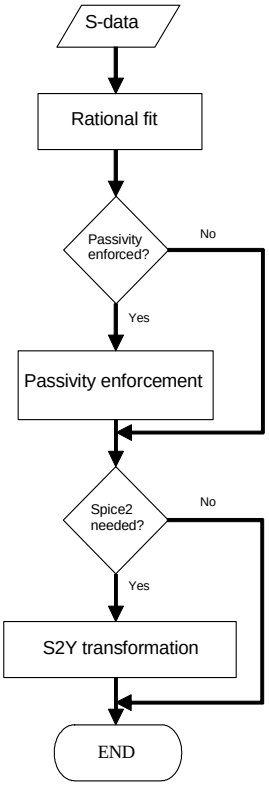


Figure 1 Broadband SPICE Model Generator Process Flow Chart

Using Broadband SPICE

Broadband SPICE State File

Broadband SPICE uses ascii *state files* (*.stf*) to save and recall tool dialog settings.

A default state file is provided with Broadband SPICE. This is used to initialize the Broadband SPICE dialogs with the default values. The default state file, named *bbs_default.stf* is located in the *$HPEESOF_DIR/bbspice/config* directory.

The default Broadband SPICE *state file* is configurable on a <project>/<user>/<site> basis like other ADS tools.

Following is an example of a Broadband SPICE application state file:

```
input.isMeasuredData 0;
model.dataSampling All;
model.targetAccuracy 5.000000e-3;
model.autoGenerate 1;
fitting.globalPolesImposed 1;
fitting.minDeltaPolesZeros -1;
fitting.maxDeltaPolesZeros 0;
fitting.maxVFIterations 5;
fitting.leastSquareMode NORMAL;
fitting.normalizeMatrix 1;
fitting.useRelaxation 0;
passivity.mirrorPolesIntoLHP 1;
passivity.enabled 1;
passivity.maxPreDataTruncation 9.999000e-001
passivity.maxNrIterations 5;
passivity.maxPostModelScaling 9.950000e-001;
output.spice2g6 0;
output.spice3 1;
output.agilentModel 1;
output.useInputDir 0;
output.path networks/;
dataset.generate 2;
dataset.path data/;
```

Starting Broadband SPICE

To start the Broadband SPICE tool select **Tools** > **Spice Model Generator** > **Start Broad Band Generator** from a *Schematic* window.

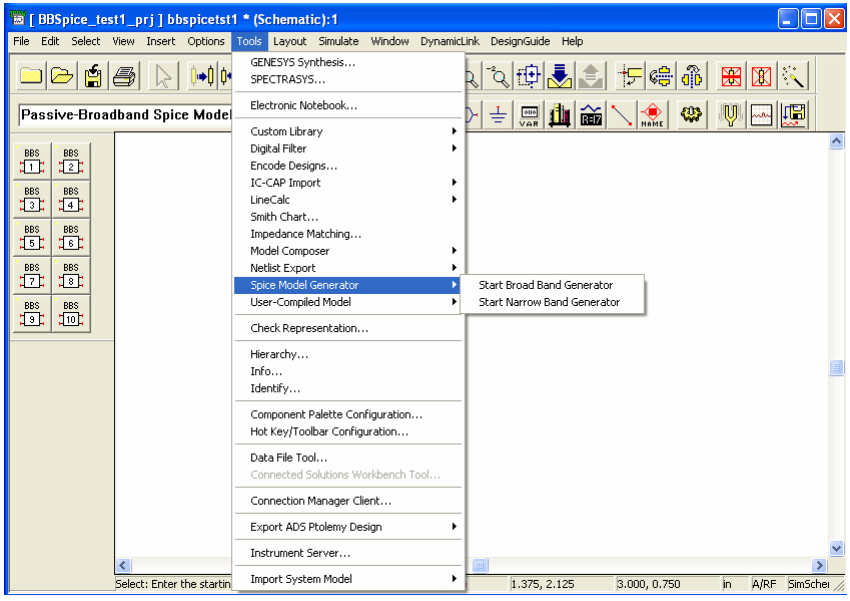


Figure 2 Starting the Broadband SPICE Model Generator.

The Broadband SPICE Main Window

This window offers three option tabs, *Standard*, *Advanced Input* and *Advanced Output*. These options are described in detail in the following sections.

In the simplest operation, it is only necessary to specify the input filename and file type before generating a model. In this case, the engine generates and uses the values for the advanced options based on its processing of the input data and the information contained in the default Broadband SPICE state file. The output file types and directory placement have been selected to easily facilitate the use of these models in ADS.

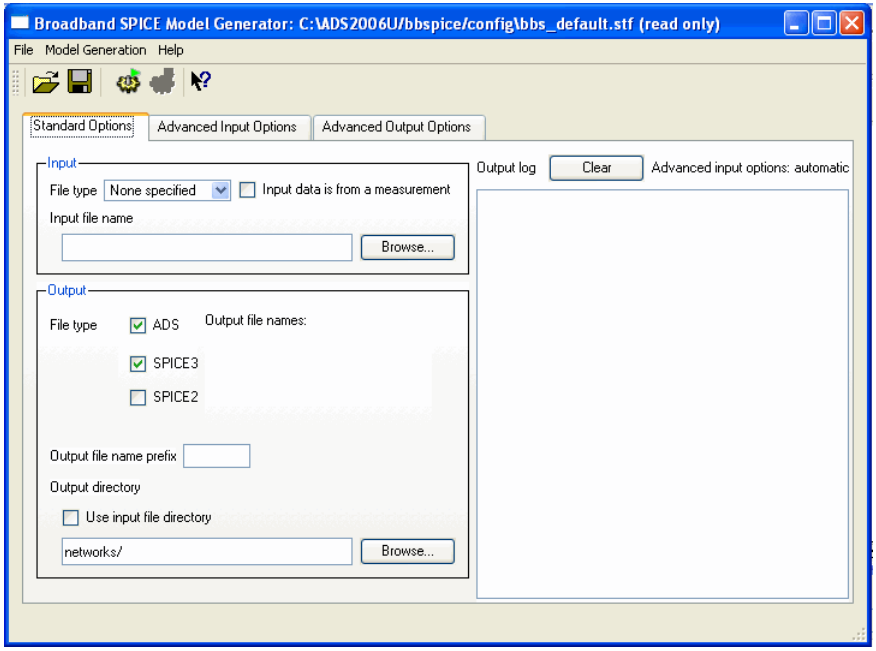


Figure 3 The Broadband SPICE Main Window

Standard Options Tab

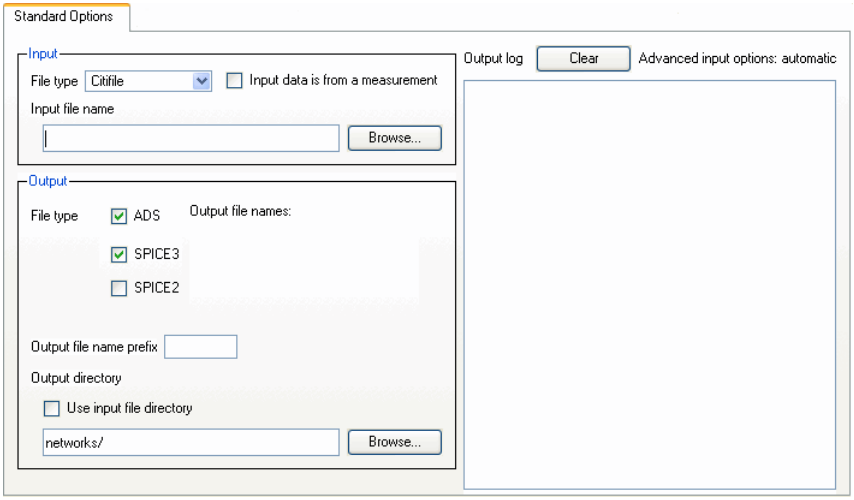


Figure 4 Standard Options Tab

The following dialogs are available under the *Standard Options* tab:

Input Dialog

The input data used by Broadband SPICE must contain S-parameter information with reference impedance of 50 Ω.

NOTE If only other network parameters are available (e.g., Y, Z, H, ABCD, etc...) an up front transformation to S-data is required.

File type The following Input files are supported by Broadband SPICE:

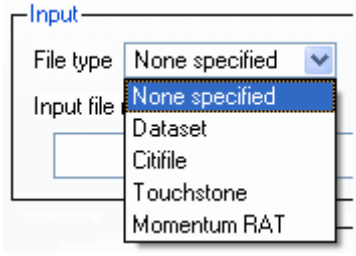
Dataset: ADS binary format

Citifile:

Touchstone:

Momentum RAT: Momentum rational (*.rat*) files are the output result of a Momentum AFS simulation. These files contain information about the rational fitting model of a single Momentum simulation and are located under the <*mom_dsn*>/<*layout_name*> directory of your project. When using this file type, you must provide the *.rat* filename (generally <*proj*>*.rat)*. It is preferred, but not required, that the *.sam* file is in the same directory as the *.rat* file. This *.sam* file contains information about the frequency samples chosen by the Momentum AFS algorithm. We recommend that you use the name of your layout as a prefix for the output model files.

NOTE Momentum is an Agilent 3-D planar electromagnetic simulator integrated into ADS and RFDE.



Input parameter data must have a reference impedance of 50 ohms.

Default: *None specified*

Input data is from measurement Specifies whether or not the input data is measured data.

A slightly different rational fitting algorithm (using relaxation) is used in model generation when this box is checked. Selecting this box is recommended if the input data is noisy. If the default fitting procedure does not converge, this option can be switched on or off.

Default: *unchecked*

Input file name Enables you to select your input file. This can be done by typing the *path* and *filename,* or by selecting the *Browse* button.

Output Dialog

This dialog enables you to select the type of models that are generated by the Broadband SPICE Model Generator tool.

File type Two model files, *<filename>.ckt* and *<filename>.bbn* are created when you select the **ADS** checkbox.

These file types are 2 different versions of the Broadband Spice model represented by ADS netlists. The *<filename>.bbn* netlist describes a polynomial representation, while the *<filename>.ckt* netlist is an ADS version of the *spice3* format.

In some cases, the generation of the rational polynomial model will fail due to numerical representation inaccuracies. In this case a warning is issued and only the *<filename>.ckt* model file is created.

NOTE

The *<filename>.bbn* and *<filename>.ckt* files are used with BBSnP circuit components in ADS schematics. For more information on using these files in ADS, refer to "Using Broadband SPICE Models in ADS" on page 44

The preferred input file type for simulations in a schematic layout using a Broadband SPICE component is *<filename>.bbn*.

If **SPICE3** is selected, a *<filename>.sp3* file is created. This file can also be used in *hspice* simulations.

If **SPICE2** is selected, a *<filename>.sp2* file is created. This format is not recommended, since it requires the engine to perform extra transformations which could diminish the accuracy of the output.

If the .s2p file is used, it is possible that a number of warnings will be issued with regard to the magnitude of the lumped elements. These warnings can be ignored.

NOTE

If you require a SPICE output, we recommended that you use the **SPICE3** format and not the **SPICE2** format.

Default: *ADS* and *SPICE3*

Output log This window displays status messages regarding various Broadband SPICE operations (e.g., model generation).

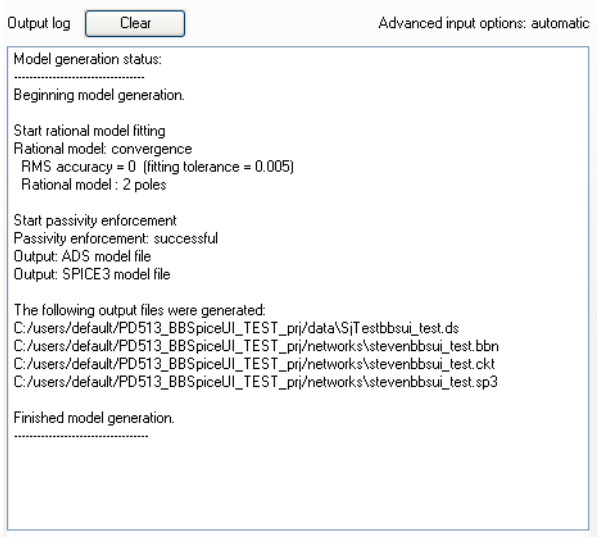


Figure 5 Output Log

To clear this window select **Clear**.

The *Advanced input options* located directly above the *Output log* indicates the status of the advanced options. It will either read *automatic* or *specified*.

If *automatic,* the input data-dependent advanced option values are automatically calculated during model generation. If *specified*, the input data-dependent advanced option values specified in the dialog will be used.

Advanced Input Options Tab

NOTE The *Advance Input Options* tab page can only be accessed after a valid file name and type have been specified on the *Standard Options* tab.

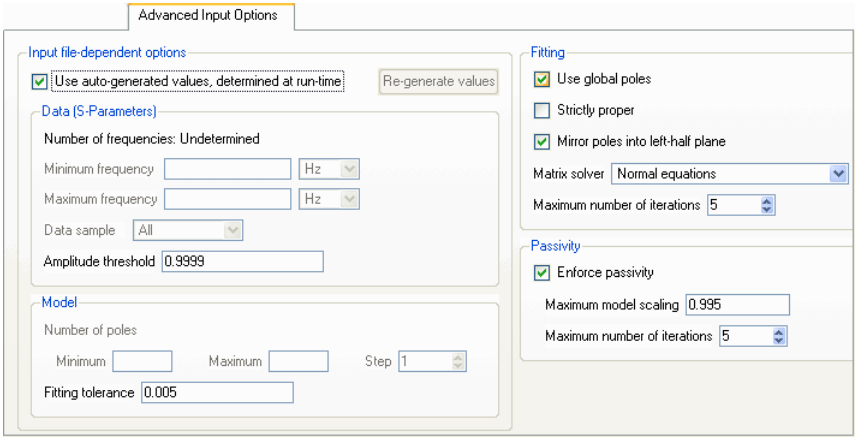


Figure 6 Advanced Options Tab

The following dialogs are available under the *Advanced Input Options* tab:

Input File-dependent Options Dialog

Certain advanced input options are highly dependent on the input data. These values are initially disabled in the dialog and cannot be modified. It is possible that model generation will fail, or will produce inaccurate results if these option values are not consistent with the input data. Because of this it is recommended that you let the engine calculate these values automatically during model generation. However, if desired, these option values can be specified manually in the dialog. The next section describes how this can be done.

Use auto-generated values determined at run-time When this checkbox is checked the Broadband SPICE engine determines the optimum values to be used, during model generation

run-time, based on the values present in the input file. When this checkbox is checked, only *Amplitude threshold* and *Fitting tolerance* values are editable.

When un-checked, the values shown in the dialog will be used during model generation, and all fields are enabled for editing. If the current values are un-initialized (as indicated by *Number of frequencies* set to "-1" or "unspecified") un-checking this checkbox will cause all dependent option values to be generated and initialized in the dialog with the recommended values.

Re-generate values Clicking this button causes all of the input file-dependent options to be re-generated and set back to their recommended values. This button is enabled only when the Use auto-generated values determined at run-time checkbox is un-checked.

Default: *Checked*

NOTE

The engine is run to initialize or re-generate the option values. This operation normally takes only a few seconds. When the operation has finished the values in the dialog are updated automatically.

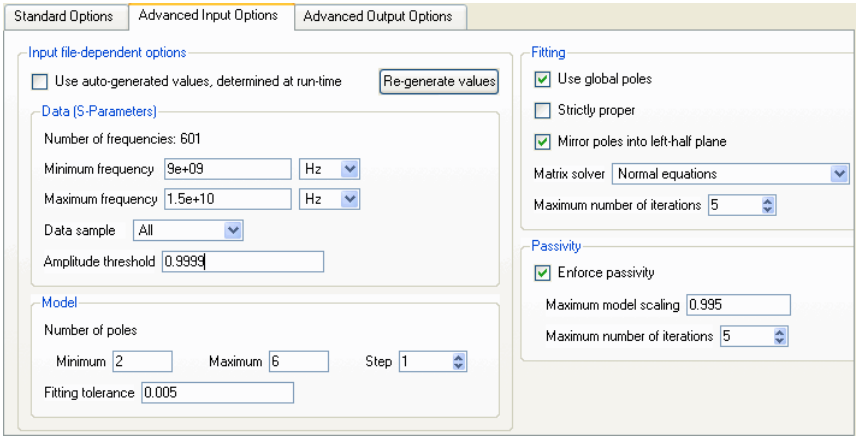


Figure 7 Re-generated Values

Data (S-Parameter) dialog

Number of frequencies This is the number of frequency points of the original input data. This value cannot be changed manually. A value of "-1" or "unspecified" indicates that the values shown in the dialog are un-initialized, and that they will be re-calculated and updated the next time either the *Use auto-generated values determined at run-time* checkbox is un-checked, or the *Re-generate values* button is pressed.

Minimum frequency and maximum frequency These values specify the minimum and maximum frequency of the input data to be used when the output models are generated. The recommended, initial values correspond to the minimum and maximum frequencies specified in the input data. When modifying these fields the following rules must be followed:

- The new minimum frequency may not be less than the minimum frequency of the input data.
- The new maximum frequency may not be greater than the maximum frequency of the input data.
- The minimum frequency may not be greater than the specified maximum frequency.

NOTE Changing these fields is not recommended.

Data sample This dropdown box specifies the number of data samples used when generating the output model, relative to the Number of frequencies (defined above) in the input data.

A distinction should be made between the different types of input files when using this option. Different file types provide better results if the following rules are applied.

- *Momentum RAT*: The recommended setting is **AllX2**

Momentum saves a limited number of data points in the *.rat* and *.sam* files. Therefore, the preferred setting is *AllX2*. However, *All* or *AllX5* may also be used.

- *Dataset, Citifile and Touchstone*: The recommended setting is **ReducedX20**

  The fields *All*, *ReducedX2*, *ReducedX5*, *ReducedX10*, *ReducedX20* and *ReducedX50* can all be selected. Reducing the number of data points may be helpful, especially if there are a large number of frequencies in the input data and the number of poles is limited. This can provide a large time gain without any loss of accuracy. If a chosen reduction removes too many data points, a warning is issued and an automatic switch to a less coarse reduction scheme is executed.

Amplitude threshold To facilitate passivity later on, the singular values of the S-parameters are upper limited to the amplitude threshold. If the singular values of the S-parameters are greater than 1, by definition, the input data is not passive (this may be caused by measurement noise or numerical simulation inaccuracies).

Limiting the max value of the input data to the specified Amplitude threshold value is a preprocess passivity enforcement step which is applied on all discrete data samples. The recommended value is 1 or slightly less than 1 (e.g., 0.9999).

If you wish to skip this preprocess step, you can specify a very large value (e.g., 100 or so). This is not recommended.

Default Value: 0.9995

Model

The input data is used to generate a rational model. This model has the form of a series expansion in partial fractions:

$$\sum_{n}^{N} \frac{r_n}{s - p_n}$$

The factors r_n are the residues, while p_n are the poles. In total, there are N such terms:

$$\frac{r_1}{s - p_1} + \frac{r_2}{s - p_2} + \cdots + \frac{r_n}{s - p_n}$$

Depending on the dynamics of the input data, the number of terms (*N*) varies.

So, the number of poles (and thus terms) is dependent on the input data. The recommended range for the number of poles is based on the analysis of the dynamics of the input data. You are free to modify this number if desired.

Number of poles: Minimum, maximum and step The recommended, initialized values are calculated based on the dynamic behavior of the input data.

If the fitting process does not converge, or if passivity enforcement fails, changing the number of poles may lead to convergence and passivity. This is especially true for noisy data, when it can be difficult to estimate the *Minimum* and *Maximum* number of poles of the system.

Step defines the sweep step within the interval [Minimum, Maximum] number of poles. It is by definition larger or equal to zero and smaller than |Maximum – Minimum| number of poles.

Fitting tolerance This is the allowed root mean square (RMS) fitting error. It is the criterion to decide if the rational fit will be successful or not. If the RMS value of the rational model fitted data is less than the fitting tolerance, the model is considered to be converged.

The recommended setting for this value ranges between *0.001* to *0.02*.

Default Value: *0.005*

Fitting

Use global poles Checking this box causes all S-parameters to share a common set of poles. This setting is advised if the number of ports is limited. If the number of ports is higher (typically above 4), un-checking this box may increase the speed of the fitting process, but significantly decrease the speed of the passivity enforcement process.

Default: *Checked*

Strictly proper If the *Strictly proper* box is checked, the following properties are valid:

N = D – 1, with **N** the polynomial degree of the numerator and **D** the polynomial degree of the denominator of the rational model.

If the box is un-checked, the rational model is called *proper* and **(D - 1)** ≤ **N** ≤**D**.

NOTE

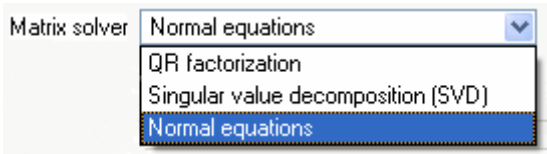
If passivity for very high frequencies is desired, selecting *Strictly proper is recommended*.

Default: *Un-checked*

Mirror poles into left-half plane This parameter forces the non-physical poles to be mirrored into the left half complex plane. Note that poles must be in the left half plane in order to obtain a passive model. If fitting convergence fails, modifying this field can lead to convergence.

Default: *Checked*

Matrix solver This field enables you to control the speed versus accuracy of the model generation. If there are many ports and data samples, *Normal equations* is the preferred setting. If accuracy is the most important criterion and speed is not essential, then *Singular value decomposition* should be selected. If none of these factors are important, and if the number of poles and ports are limited *QR factorization* may be used.



Default: *Normal equations*

Maximum number of iterations (Fitting) This is the number of pole relocation steps (≥1). The higher this value, the more time it can take for model generation. Values between 4 and 7 are recommended.

Default: *5*

Passivity

Enforce passivity This enables you to select whether the passivity of the rational model will be checked or imposed.

If passivity enforcement is not selected, the output files are generated based on the rational fitting model. If passivity enforcement is selected, an additional post-processing step is performed, which can modify the coefficients of the fitting model to ensure passivity. After this step is executed, the output files are generated based on the (passivity enforced) rational fitting model.

Default: *Selected*

Maximum model scaling This is an optional scaling factor, only meant to suppress minor (remaining) passivity violations. For example, if a maximum passivity violation of 1.0001 remains after passivity enforcement, a scaling of the broadband model by 0.9999 will resolve all remaining passivity issues.

It is the very last step in a series of passivity enforcement actions and equally affects all frequencies (from DC to infinity). The *Maximum model scaling* is the lower end of the scaling factor allowed to be used to enforce passivity.

If the you want to skip this post processing step, specify an input a value ≥ 1 in this field.

The recommended setting is slightly less than 1 (e.g., 0.999).

Default Value: *0.995*

Maximum number of iterations (Passivity) This is the number of passivity enforcement steps.

If the value is set to 0, passivity is not enforced, although a passivity check is performed on the rational fit model.

Default: *5*

NOTE

If passivity enforcement is not selected, all output files will contain the rational fitting model of the input data. If passivity enforcement is selected, all output files contain the passive rational fitting model of the input data.

Advanced Output Options Tab

NOTE The *Advanced Output Options* tab can only be accessed once a valid file name and file type have been specified on the *Standard Options* tab page.

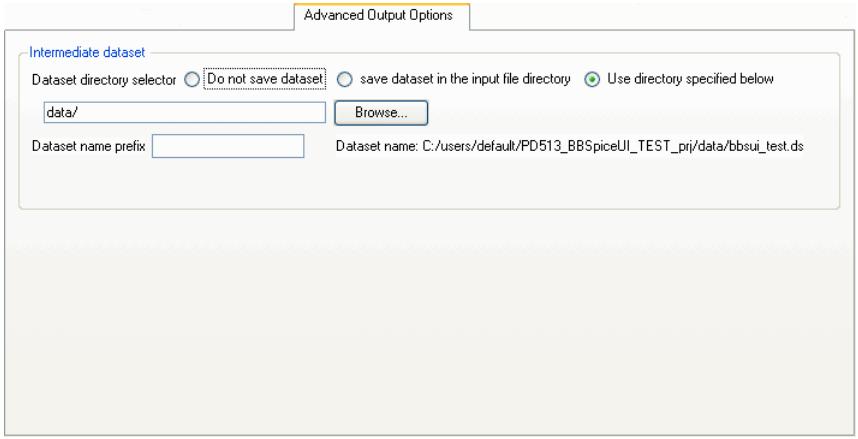


Figure 8 Advanced Output Options Tab

The following dialog is available under the *Advanced Output Options* tab.

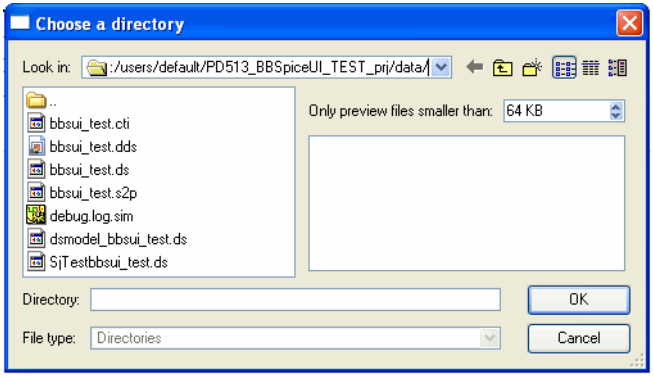
Intermediate dataset

This dialog enables you to select whether or not to save a dataset of the rational model, select a prefix name for this dataset and choose a directory in which to save it.

Data directory selector Using this dialog you can choose to:

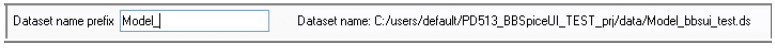
- Not save a data set
- Save the dataset in the input file directory
- Use a specified directory

  If you select the *Use directory specified below* radial button, you can supply a directory path directly, or use the *Browse...* button to find the desired location. This path defaults to the current projects data directory.



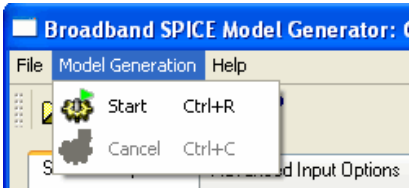
Dataset name prefix This enables you to choose a unique name for the output dataset.

Dataset name Lists the current path and dataset name to be used for the output file. When a new path is selected the current path and dataset name are automatically updated.



Broadband SPICE Model Generation

Once the model generation options have been specified, you may begin model generation by either selecting **Model Generation** > **Start** from menu bar on the main *Broadband Spice Model Generator* window or by choosing the *Start Model Generation* icon  from the main toolbar.



2
Creating Broadband SPICE Models

This chapter illustrates how to use the Broadband SPICE Model Generator tool to create broadband spice models for use in ADS and other circuit simulators.

Creating Models with Broadband SPICE Model Generator

The Broadband SPICE Model Generator enables you to easily create broadband SPICE models for use in ADS and other simulators. The following section offers an example of model creation using a single via passive component (based on full-wave EM field simulation data) and another example of coupled microstrip lines (with input from measurements).

Broadband SPICE Model Generation Examples

Overview

The following examples, show how to use the Broadband SPICE Model Generator tool to create broadband SPICE models for 2 passive circuit components based on S-Parameter data (obtained through either measurements or simulations). Later in Chapter 3, "Using Broadband SPICE Models" the generated broadband SPICE models will be used in an ADS frequency-domain and time-domain circuit simulation. The initial S-Parameter data will be compared with the generated broadband SPICE models. It will be shown that the Broadband SPICE Model Generator tool successfully produced accurate, broadband models of the S-Parameter data, and that these broadband SPICE models can be easily used in the ADS circuit simulators.

NOTE The data used for these examples was obtained from *Wendemagegnehu T. Beyene from Rambus Inc.*

Input Data

The SPICE Model Generator tool can use S-Parameter data files, in several standard file formats, as inputs.

NOTE The input data can be either measured or simulated.

Model Creation Using a Package Via Structure

This example uses S-Parameter data obtained from passive circuit elements as the input data for the broadband SPICE model generation.

The example file is located in the Agilent EEsof Knowledge Center at:

https://edasupportweb.soco.agilent.com/cgi-bin/show.pl?id=275600

Select **Software Downloads** > **ADS** > **ADS 2006** > **ADS 2006 Update 1** > **BBS_Training**

The input data for this example is from a package single via structure. The 2-Port S-Parameter data, in *Touchstone* file format, was obtained from simulations using a full-wave (3D) electro-magnetic field solver.

Input touchstone file name: *SingleVia.s2p*

Equivalent dataset file name: *REF_SingleVia.ds*

The following section illustrates how to use the ADS Broadband SPICE Model Generator tool to create an accurate SPICE model from the input S-Parameter data.

1. Start **ADS.**
2. Select **File** > **Unarchive Project...** and unarchive the file: *BBS_Training.zap*

NOTE This project file contains the input data files to be used in this example during broadband SPICE model generation, as well as, schematics and data displays that will be used to verify the results.

3. Open a *Schematic* window
4. From the *Schematic* window main menubar, select **Tools** > **Spice Model Generator** > **Start Broad Band Generator** to open the Broadband SPICE Model Generator tool.

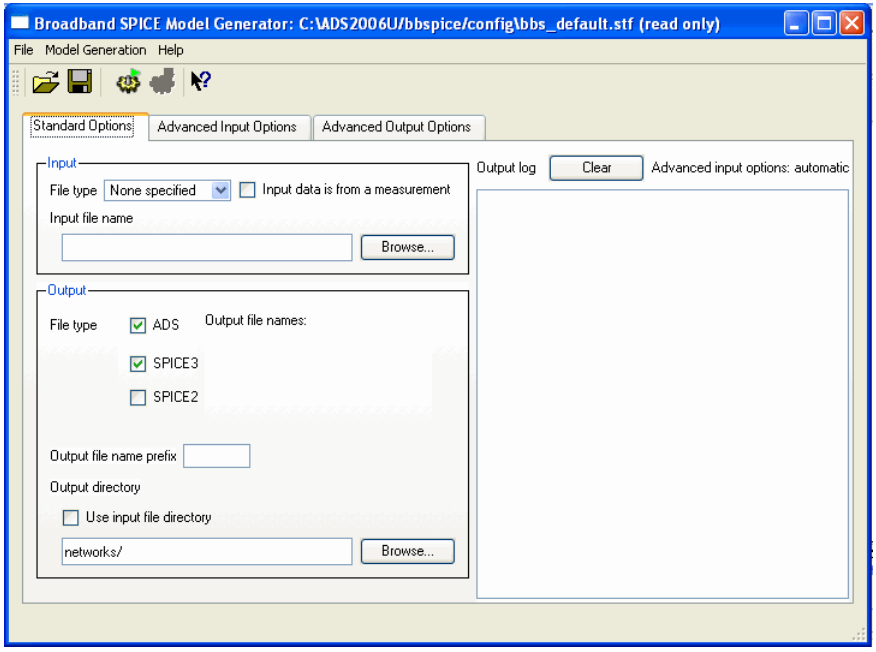
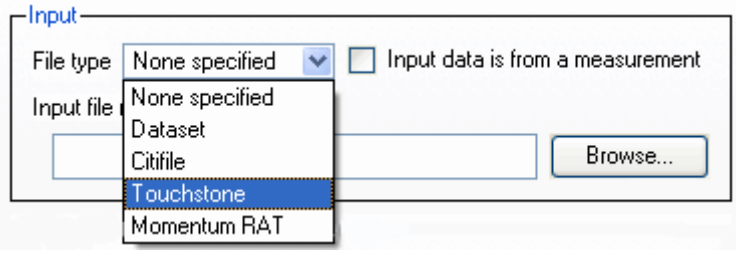


Figure 9 Broadband SPICE Model Generator Main Window

Setting up the Broadband SPICE Model Generation Tool

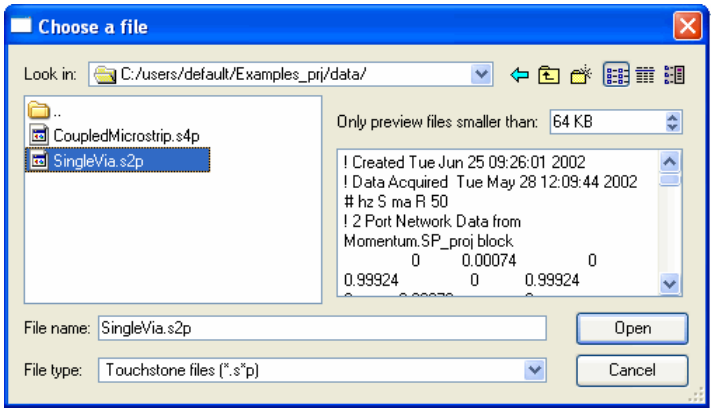
This section explains how to set up the tool to generate broadband SPICE models.

1 From the *Input* section of Broadband SPICE Model Generation Tool *Standard Options* tab, select **Touchstone** in the *File type* dropdown menu



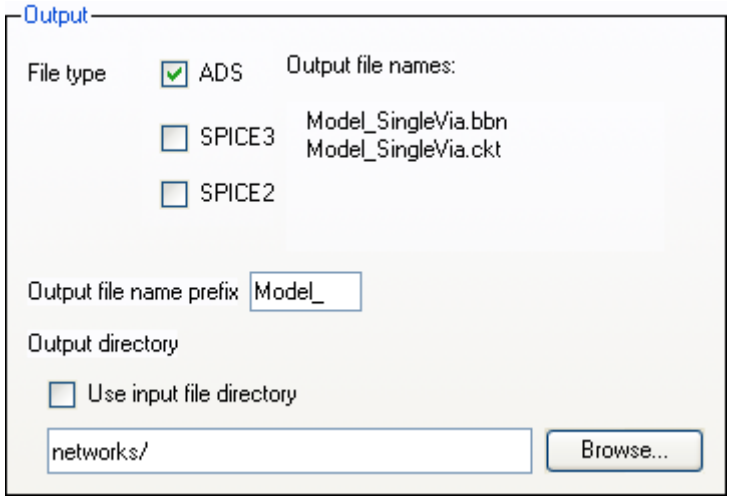
NOTE You must select a *File type* from in the *Input* section to access the *Advanced Input* and *Advanced Output* options tab

2 Click the **Browse** button associated with *Input File Name* dialog and select **SingleVia.s2p** from the *Browse* window.

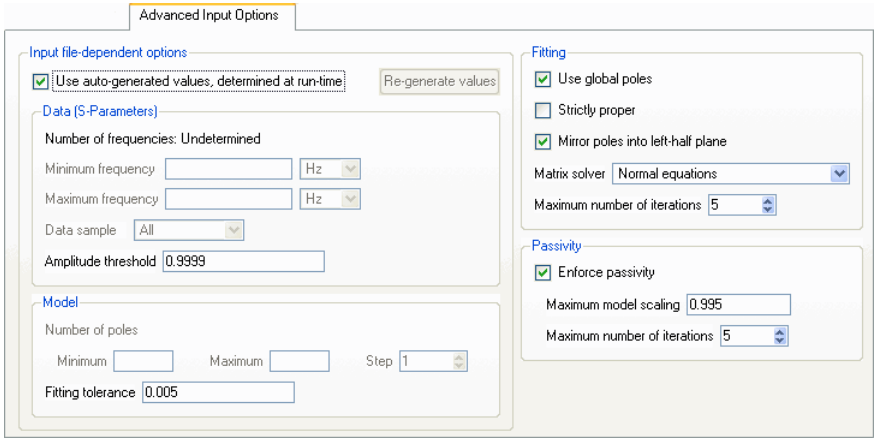


3 In the *Output* section, check **ADS** as the *Output File Type*.

4 Type "**Model_**" in the *Output file name prefix* dialog box.



5 Next, click the **Advanced Input Options** tab.

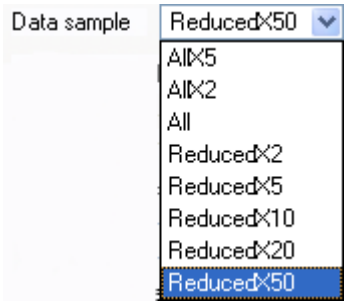


6 Un-check the **Use auto-generated values, determined at run-time** check box.

Un-checking this box causes the engine to generate recommended initial values for the input data-dependent options, and activates the deactivated selections in the *Independent file-dependent* options section.

NOTE The engine is run to initialize or re-generate the option values. This operation normally takes only a few seconds. When the operation has finished the values in the dialog are updated automatically.

7 In the *Data Sample* section of this window, select **ReducedX50** from the dropdown menu



8 Click the **Advanced Output Options** tab.

9 Ensure that the *Dataset name prefix* is set to **Model_**.

NOTE The only *Advanced Option* setting having an affect on the output model calculation, changed from its recommended value for this example, was *Data sample*.

Broadband SPICE Model Generation

Once the *Standard*, *Advanced Input* and *Advanced Output* options have been set, you are ready to begin the model generation process.

1 To begin model generation, select **Model Generation** > **Start** from the Broadband SPICE Model Generation main menu.

Note that once the model generation has started, the tab automatically switches to the *Standard Options* page. This enables you to review the model generation status and view other information available in the *Output log* dialog.

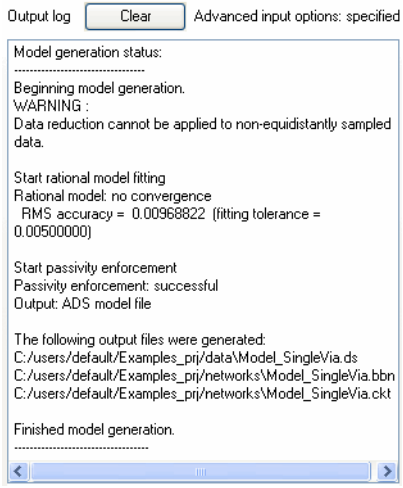


Figure 10 Output Log

For this example 3 output files are generated:

- *<output_dir>/Model_SingleVia.bbn*
- *<output_dir>/Model_SingleVia.ckt*
- *<dataset_dir>/Model_SingleVia.ds*

2 Model creation is now complete. To exit the Broadband SPICE Model Generation Tool, select **File** > **Exit**

3 Click **No** if the *Save Changes* dialog that appears.

4 Exit **ADS**.

This completes the example for using the ADS Broadband SPICE Model Generator tool to create broadband SPICE models (*.bbn* and *.ckt*) for a single via passive circuit element.

For information on how to use and verify these models in ADS, refer to, "Using Broadband SPICE Models in ADS" on page 44 of this manual.

Model Creation Using an FR4 Coupled-Microstrip Lines Circuit

This example uses S-Parameter data obtained from passive circuit elements as the input data for the broadband SPICE model generation.

- The input data for this example was obtained from 2 6-inch long FR4 coupled-microstrip lines circuit and 4-Port S-Parameter data, in touchstone file format, obtained from Vector Network Analyzer measurements.[1]

  Input touchstone file name: *CoupledMicrostrip.s4p*

  Equivalent dataset file name: *REF_CoupledMicrostrip.ds*

The following section illustrates how to use the ADS Broadband SPICE Model Generator tool to create an accurate SPICE model from the input S-Parameter data.

1 Start **ADS.**

2 Open **BBS_Training_prj**.

NOTE

This project file contains the input data files to be used in this example during broadband SPICE model generation, as well as, schematics and data displays that will be used to verify the results.

3 Open a *Schematic* window

4 From the *Schematic* window main menubar, select **Tools** > **Spice Model Generator** > **Start Broad Band Generator** to open the Broadband SPICE Model Generator tool.

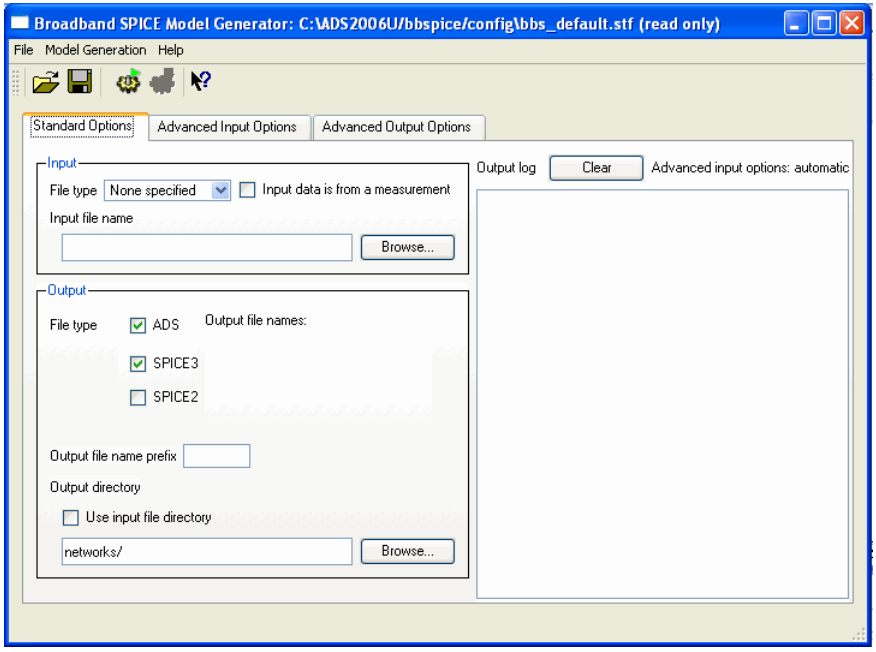


Figure 11 Broadband SPICE Model Generator Main Window

Setting up the Broadband SPICE Model Generation Tool

This section explains how to set up the tool to generate broadband SPICE models.

1 From the *Input* section of Broadband SPICE Model Generation Tool *Standard Options* tab, select **Touchstone** in the *File type* dropdown menu.

NOTE You must select a *File type* from in the *Input* section to access the *Advanced Input* and *Advanced Output* options tab

2 Click the **Browse** button associated with *Input File Name* dialog and select **CoupledMicrostrip.s4p** from the *Browse* window.

3 In the *Output* section, check **ADS** as the *Output File Type*.

4 Type "**Model_**" in the *Output file name prefix* dialog box.

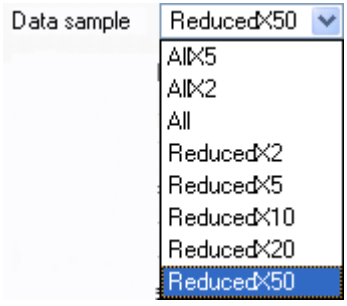
5 Next, click the **Advanced Input Options** tab.

6 Un-check the **Use auto-generated values, determined at run-time** check box.

Un-checking this box causes the engine to generate recommended initial values for the input data-dependent options, and activates the deactivated selections in the *Independent file-dependent* options section.

NOTE The engine is run to initialize or re-generate the option values. This operation normally takes only a few seconds. When the operation has finished the values in the dialog are updated automatically.

7 In the *Data Sample* section of this window, select **ReducedX50** from the dropdown menu



8 Click the **Advanced Output Options** tab.

9 Ensure that the *Dataset name prefix* is set to **Model_**.

NOTE The only *Advanced Option* setting having an affect on the output model calculation, changed from its recommended value for this example, was *Data sample*.

Model Generation for the FR4 Coupled-Microstrip Lines Circuit

Once the *Standard*, *Advanced Input* and *Advanced Output* options have been set, you are ready to begin the model generation process.

1 To begin model generation, select **Model Generation** > **Start** from the Broadband SPICE Model Generation main menu.

Note that once the model generation has started, the tab automatically switches to the *Standard Options* page. This enables you to review the model generation status and view other information available in the *Output log* dialog.

2 Check the *Output log*.

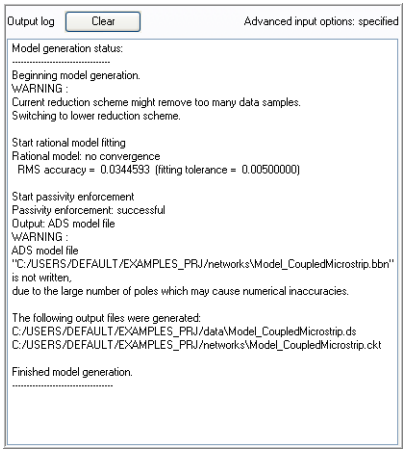


Figure 12 Output Log

Notice that during the simulation run, the status window indicates the rational model did not converge (an RMS error of 0.03446 was obtained, while the default RMS tolerance is 0.005). You should also notice that *Passivity enforcement* was successful.

One possible way to improve RMS accuracy is to use the *Input data is from measurement* option.

3 On the *Standard Options* tab check the **Input data is from measurement** checkbox. Leave all other settings as they were.

4 Re-run the simulation.

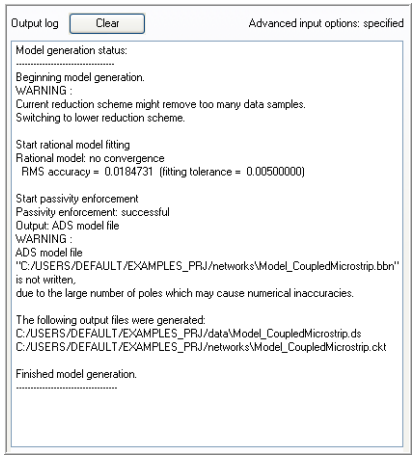


Figure 13 Improved convergence, using the *Input data is from measurement* option.

The *Output Log* now shows an RMS accuracy of 0.01847, which approximately two times better than the previous simulation.

NOTE If the accuracy of the rational model is insufficient, it is possible that selecting the *Input data is from measurement* option will improve the accuracy. This is independent of whether or not the data is from an actual measurement.

For this specific simulation, the model generation time took much longer than in the first example. This is due to the higher number of ports and the strong dynamical behavior of the input data, which leads directly to a high number of poles. If the passivity of the model is of minor importance, the execution time of the engine can be

decreased importantly by not checking the *Enforce passivity* parameter in the *Passivity* box of the second tab *Advanced Input Options*.

If passivity is not important, but if you'd like to know whether the rational fitted model is passive, you can select the *Enforce passivity* parameter and set the *Maximum number of iterations* to zero. In this example the rational fitting model was already passive.

If passivity of the model is not important, the fastest way to obtain a model is to uncheck the *Use global poles* option in the *Fitting* box of the *Advanced Input Options* tab.

In this example only 2 output files were generated:

- <*output_dir*>/*Model_CoupledMicrostrip.ckt*
- <*dataset_dir*>/*Model_CoupledMicrostrip.ds*

In this example the *.bbn* file was not generated. This is because as the number of poles increase, numerical inaccuracies can occur. However, the *.ckt* file can still be used in an ADS schematic to obtain the desired results. It should be noted that if both ADS model files are written, the *.bbn* file is the preferred file type.

5 This completes the Model creation. Select **File** > **Exit** to exit the Broadband SPICE Model Generation tool.

6 Click **No** if the *Save Changes* dialog that appears.

7 Exit **ADS**.

This completes the examples for using the ADS Broadband SPICE Model Generator tool to create broadband SPICE models (*.bbn* and *.ckt*).

For information on how to use and verify these models in ADS, refer to, "Using Broadband SPICE Models in ADS" on page 44 of this manual.

References

1 "*An Accurate Transient Analysis of High-Speed Package Interconnects Using Convolution Technique*" by Wendemagegnehu T. Beyene and Chuck Yuan, Rambus Inc. Los Altos, CA 94022, U.S.A. in *Analog Integrated Circuits and Signal Processing*, 35 pages 107-120, 2003.

3
Using Broadband SPICE Models

This chapter explains how to use the models created with the Broadband SPICE Model Generator in ADS.

Using Broadband SPICE Models in ADS

Verifying the Rational Fitting Model Data for the SingleVia Structure

Overview

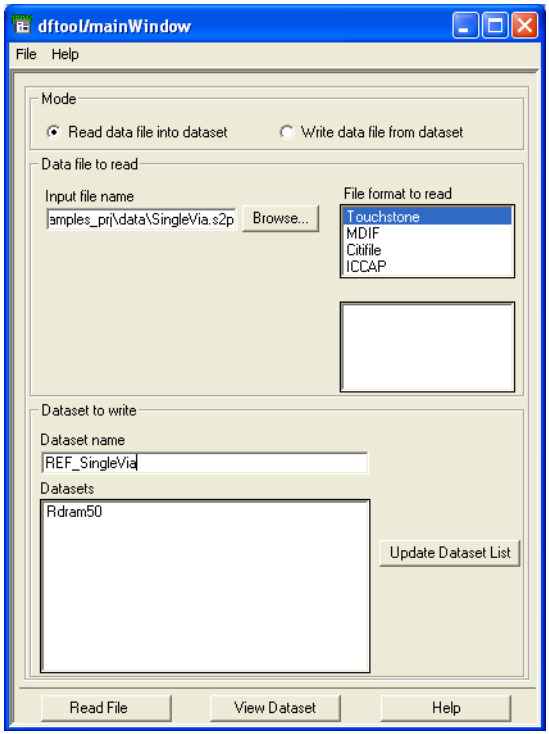
Previous examples showed how to create models using the Broadband SPICE Model Generator. In this chapter we will offer some explanation on how to use these models in ADS.

To begin with we will show how to verify the Broadband SPICE rational model data:

1. Start **ADS**.
2. Open a *Schematic* window, from the ADS Main windows menubar choose **Window** > **New Data Display**, to open a *Data Display* window.
3. From the Data Display's main menubar select **Tools** > **Data File Tool**.
4. From the Data Files Tools main window, select the radial button for "*Read data file into data set"*. Select **Touchstone** as the "*File format to read*", browse to *"SingleVia.s2p"* (the model generation input data) and type **REF_SingleVia** into the field provide for the dataset name.

5 Click the **Read File** button to finish the process.

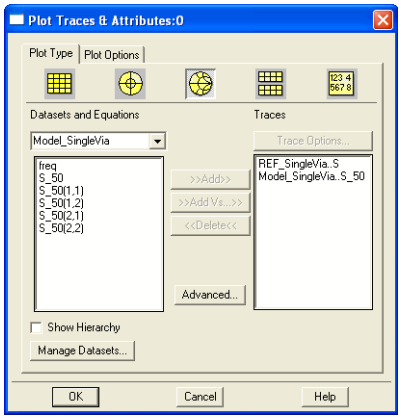
6 Close the *Data File Tool*.



7 From the Data Display window Select the Smith Chart icon  from the left toolbar to insert a Smith Chart plot onto the Data Display window.

8 Insert 2 traces onto the Smith Chart:

- REF_SingleVia.S
- Model_SingleVia.S_50



NOTE *Model_SingleVia.ds* is the dataset containing the rational fitting model of the *SingleVia* data created and saved during the model generation in the first example.

The following plot should appear in the *Data Display* window:

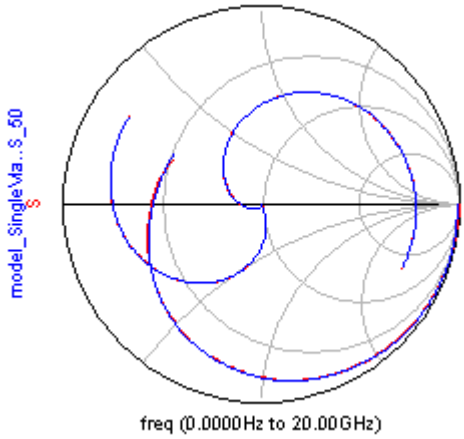


Figure 14 Input and Output Comparison of S-Parameters using a Smith Chart Plot.

The red trace plots the input data, and the blue trace plots the generated, rational model data. This plot demonstrates that there is no noticeable difference between the two and that the generated rational model is an accurate representation of the input model data.

9 Close the *Data Display* window.

Verifying the Output Model for the SingleVia Structure

1 Start **ADS.**

2 Open a Schematic window, select **File** > **Open Design**. Locate and open the design *SingleViaFreqdomain.dsn*

3 The following circuit should appear in the Schematic window.

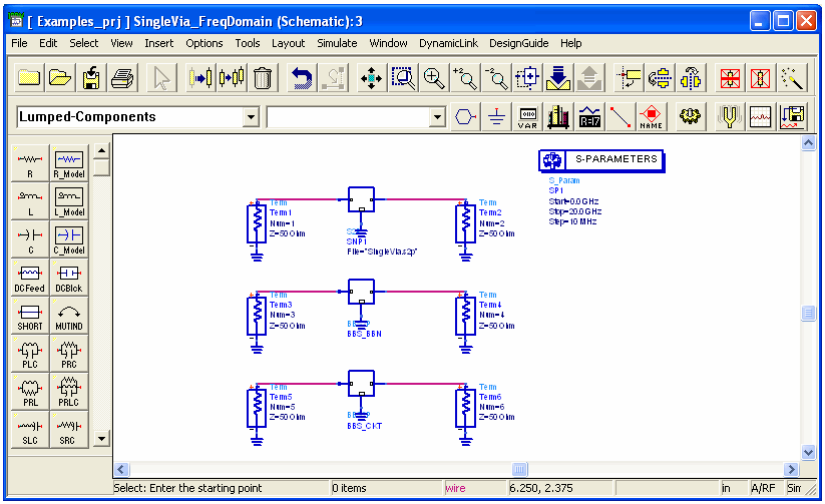
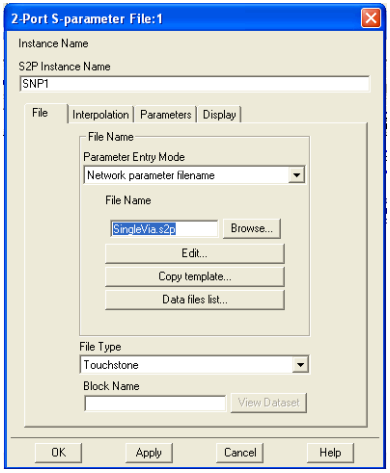


Figure 15 SingleViaFreqdomain.dsn

4 Double-click the SNP data item to bring up the *Edit Parameter* window.



5 Click the *Browse* button with the Touchstone file type selected.

6 Select the file *SingleVia.s2p*, then click **Open** and **OK**.

7 Double-click the BBS2P component BBS_BBN to bring up the *Edit Parameter* window.

8 Click the *Browse* button with the *ModelFile* parameter selected.

9 Select the file *Model_SingleVia.bbn*, click **Open** in the *Select File* window and **OK** in the *Edit Parameter* window.

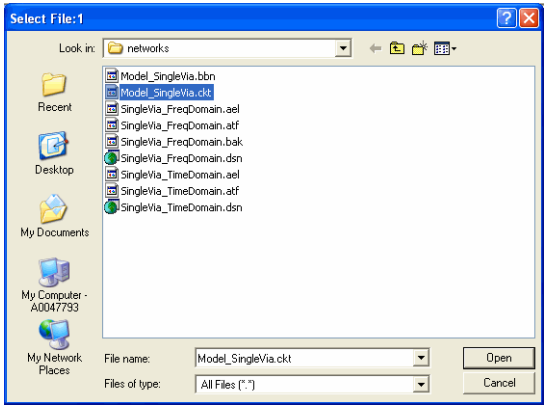
NOTE

Model_SingleVia.bbn is the ADS broadband SPICE output model file that was created and saved during the Broadband SPICE Model Generation. The BBS2P component in the schematic now represents this model, which will be used when the frequency circuit simulation is run.

10 Double-click the BBS2P component BBS_CKT to bring up the *Edit Parameter* window.

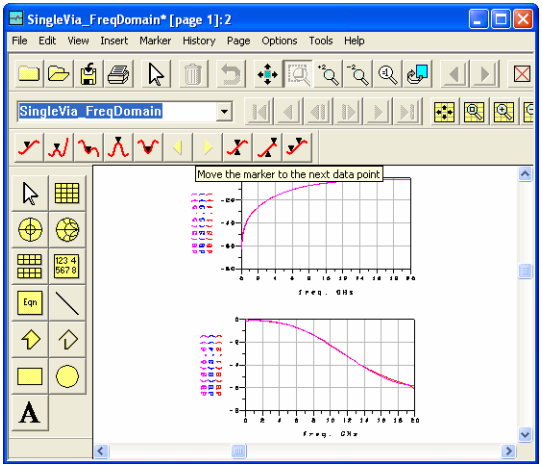
11 Click the *Browse* button with the *ModelFile* parameter selected.

12 In the *Select File* window, set the filter to All Files (*.*). Then select the *Model_SingleVia.ckt* file.



13 Select **Simulate** > **Simulate** from the Schematic window main menubar to perform the simulation.

When the simulation finishes, a Data Display window is opened: *SingleVia_FreqDomain.dds*.



From the schematic representation in Figure 15 on page 48, it is clear, for example, that S(1,1) should be equal to S(3,3) and S(5,5). This confirms the correctness of the ADS models and shows that the broadband SPICE generated output models are an accurate representation of the input model.

Using the SingleVia Structure Model in the Time Domain

1 In a Schematic window, select **File** > **Open Design**. Locate and open the design *SingleVia_TimeDomain.dsn*

2 The following circuit should appear in the schematic window.

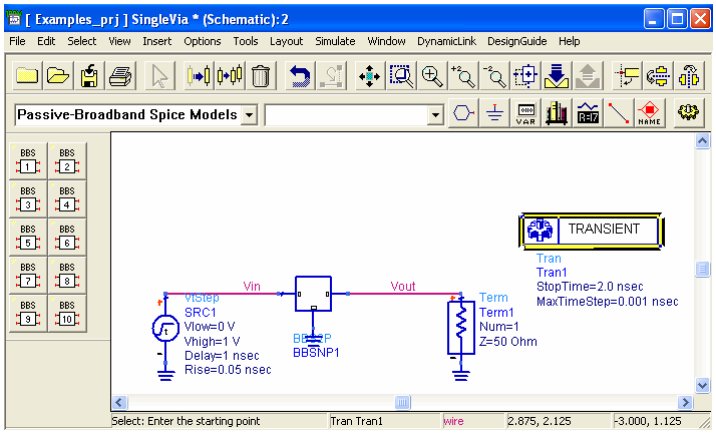


Figure 16 Transient Simulation of a Broadband SPICE component

A transient simulation of the model represented by the BBS2P component has been set-up in this schematic. The following steps will ensure that the BBS2P component is set-up to correctly represent the Broadband SPICE model generated in Chapter 2.

3 Double-click the **BBS2P component**. To bring up the *Edit Parameter* window.

4 Click **Browse** button with the *ModelFile* parameter selected.

5 Select the file *Model_SingleVia.bbn*, click **Open.**

NOTE *Model_SingleVia.bbn* is the ADS broadband SPICE output model file created and saved during the Broadband SPICE Model Generation.

6 Click **OK** to dismiss the *Edit Parameters* dialog.

7 Select **Simulate** > **Simulate** from the main Schematic window to perform the transient simulation.

8 When the simulation has completed select **Window** > **Open New Data Display.**

9 From the Data Display main menu select **File** > **Open...**

10 Select the file *SingleVia_sharp_pulse_response.dds* and click **Open**.

The following plot should appear in the Data Display window.

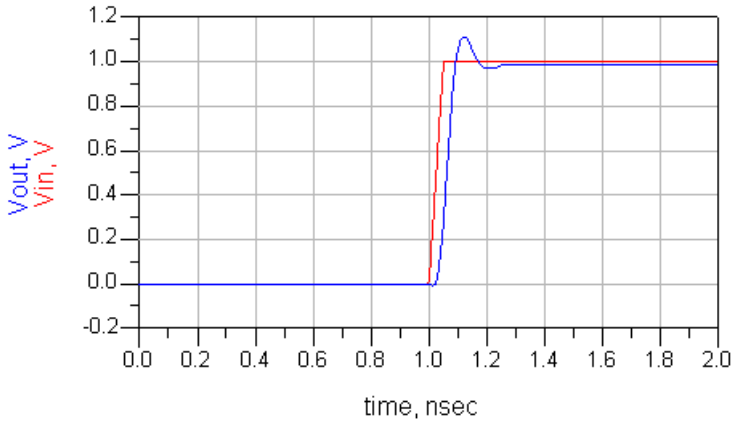
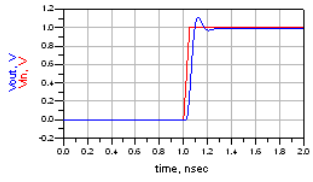


Figure 17 Transient Simulation Plot of BBS2P

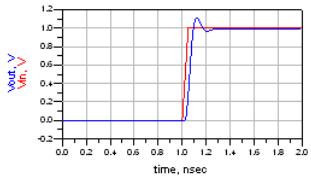
NOTE This plot shows the results of the transient analysis of the generated SPICE model. It is the voltage at nodes *Vin* and *Vout* vs. *time*.

11 To verify that the other ADS model (which makes use of controlled sources) of the broadband SPICE output model works as well as the rational polynomial model in ADS transient simulation, change the *BBS2P ModelFile* parameter value from *Model_SingleVia.bbn* to

Model_SingleVia.ckt and re-run the simulation. The results are illustrated in the following plots:



Model_SignalVia.bbn



Model_SignalVia.ckt

This demonstrates how easily Broadband SPICE generated output can be used in ADS transient circuit simulations.

Verifiying the Rational Fitting Model for the CoupledMicrostrip Lines

You can verify the rational model data and output ADS models for the CoupledMicrostrip lines example by following the same procedures outlined in, “Verifying the Rational Fitting Model Data for the SingleVia Structure” on page 44 and “Verifying the Output Model for the SingleVia Structure” on page 48.

Using the CoupledMicrostrip Lines Model in the Time Domain

1 In a Schematic window, select **File** > **Open Design**. Locate and open the design *Coupledmicrostrip_TimeDomain.dsn*

2 The following circuit should appear in the schematic window.

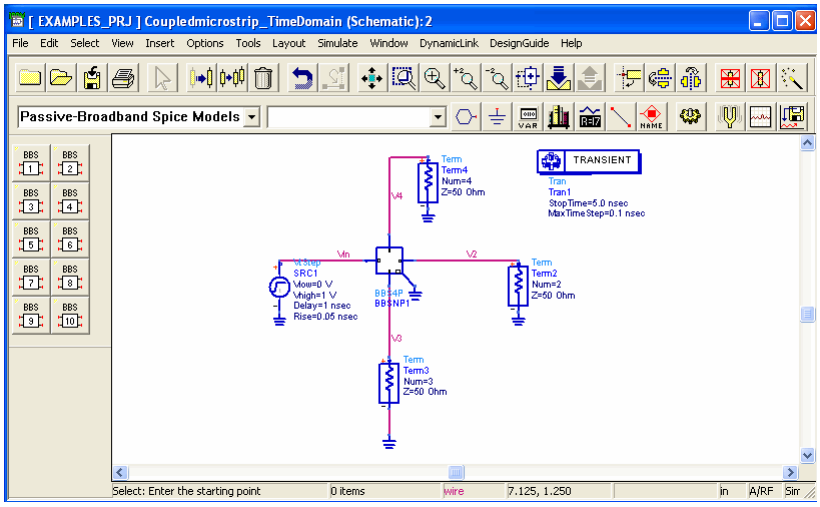


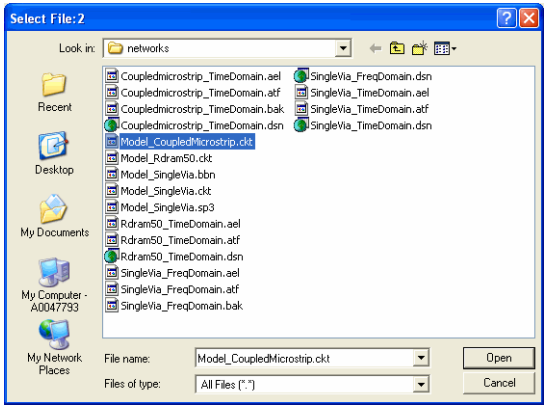
Figure 18 Transient Simulation of the Broadband SPICE Component

A transient simulation of the model represented by the BBS4P component has been set-up in this schematic. The following steps will ensure that the BBS4P component is set-up to correctly represent the Broadband SPICE model generated in the example "Model Creation Using an FR4 Coupled-Microstrip Lines Circuit" on page 35.

3 Double-click the **BBS4P component**. To bring up the *Edit Parameter* window.

4 Click **Browse** button with the *ModelFile* parameter selected.

Since the *.bbn* file was not created for this model (refer to, "Model Generation for the FR4 Coupled-Microstrip Lines Circuit" on page 38) we will use the .ckt file instead. Because of this, "*All Files*" will need to be selected in order to be make the .ckt file to selectable.



5 Select the file *Model_Coupledmicrostrip.ckt*, click **Open.**

NOTE *Model_Coupledmicrostrip.ckt* is the ADS broadband SPICE output model file created and saved during the Broadband SPICE Model Generation.

6 Click **OK** to dismiss the *Edit Parameters* dialog.

7 Select **Simulate** > **Simulate** from the main Schematic window to perform the transient simulation.

8 When the simulation has completed the *Data Display* window will open and display:

Model_Coupledmicrostrip_sharp_pulse_response.dds

The following plot should appear in the *Data Display* window:

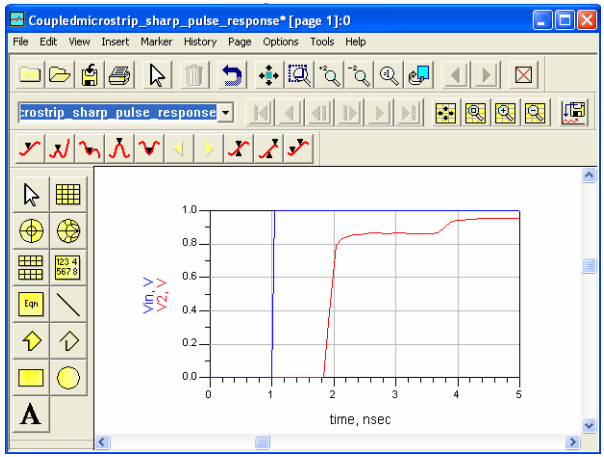


Figure 19 Transient Simulation Plot of BBS4P

NOTE This plot shows the results of the transient analysis of the generated SPICE model. It is the voltage at nodes *Vin* and *Vout* vs. *time*.

It is clear that in this example, that there is a significant delay due to the length of the line. As you can see on the figure, the delay is about 0.9 ns. You can also see the ringing effect, which confirms the 0.9 ns travel time.

Recommendations and Tips

The following information may be useful, when using the Broadband SPICE Model Generation tool:

- If you have difficulties obtaining the required convergence accuracy or passivity level, we recommend that you to check the *Input is from a measurement* option on the *Standard Options* tab first.
- If the number of poles are too high, the *.bbn* ADS model file will not be created, in most cases. Should this occur, the *.ckt* file can be used in its place. However, if the *.bbn* file is available, we recommend that you use that file.
- Unless absolutely necessary, *do not* select *.sp2* type as an your output file. The generation of this file demands extra calculation efforts and has lower numerical accuracy than the ADS model files and the *.sp3* type.
- When using *Advanced Input Options*, we recommend selecting the *ReducedX50* value in the *Data sample* section, especially if the number of frequencies is high. The engine automatically switches to a lower reduction scheme if more data samples are needed to determine a correct model. Note that the *ReducedX* field can not be applied to examples where the input file is a Momentum rational file type. For Momentum rational files, the *AllX* field should be used.
- The number of poles indicated in the *Model* section of the *Advanced Input Options* tab is an estimation based on the input data. This value can be changed, if you wish, to view the effect on the generated model.

  Generally, the estimated range of poles is adequate, however, you can adjust these ranges in order to obtain better results. This is especially the case when:

  - The input data is very noisy, making it hard to estimate the dynamical behavior.

- You have significantly modified the minimum and maximum frequencies. In this case, the number of poles will have to be adapted to the reduced frequency range. It is generally acceptable to set a wide range for the *Minimum* and *Maximum* number of poles if the correct number is not known.

- If there are too many ports and poles, simulation time can grow significantly. Unless required, skipping the passivity enforcement step can reduce the overall simulation time.

  Since the passivity enforcement calculation step can take a large portion of the total model generation time, consider the following:

  - If passivity enforcement is not needed and the a passive rational fitted model in not required, it is best to uncheck the *Use global poles* option in the *Fitting* section of the *Advanced Input Options* tab.
  - If passivity enforcement is not needed, but you want to know whether the fitted model is passive (without modifying the model to obtain passivity), you can check the *Enforce passivity* option, but set the *Maximum number of iterations* in the *Passivity* section to 0.
  - If passivity enforcement is needed, you must set *Enforce passivity* option and select a number for the *Maximum number of iterations*, typically 5.

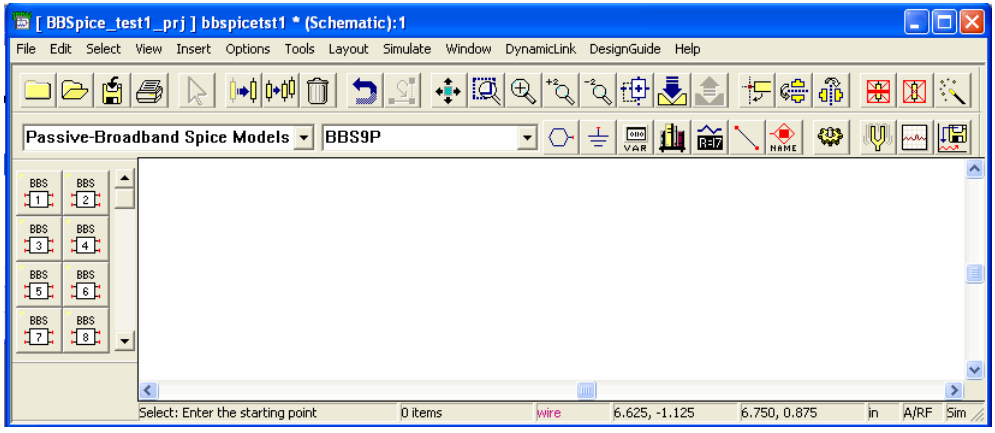
If the input data contains a large, dynamical system with many ports (and thus a large simulation time), it is a good practise to run several simulations using all of the above settings (and any other combinations, as you prefer) then save the output files with a different file name prefix, in order to compare the results using the *Data Display*.

A
Passive Broadband SPICE Model Reference

This appendix covers Passive Broadband SPICE model symbols, parameters and notes.

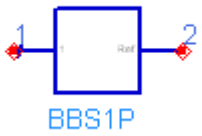
Overview

The Passive Broadband SPICE built-in components are available in the Schematic view of ADS under Passive-Broadband Spice Models section.



BBS1P(1-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 1-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.bbn (rational-polynomial formulation) and <*filename*>.*ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP

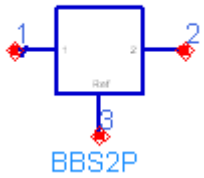
component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (2) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS2P (2-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 2-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP

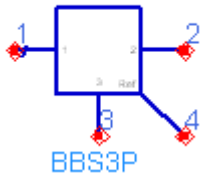
component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (3) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS3P (3-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 3-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP

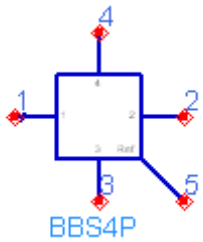
component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (4) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS4P (4-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 4-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.bbn (rational-polynomial formulation) and <*filename*>.*ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension .*bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (.*ckt*) model may provide better results.

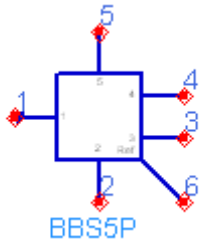
4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (5) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS5P (5-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 5-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.bbn (rational-polynomial formulation) and <*filename*>.*ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension .*bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (.*ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the

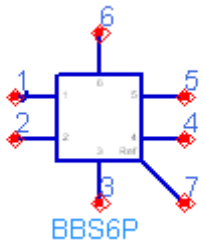
model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (6) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS6P (6-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 6-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.*bbn* (rational-polynomial formulation) and <*filename*>.ckt (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the

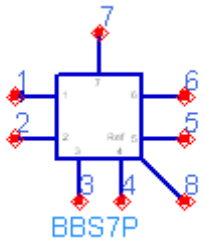
model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (7) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS7P (7-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 7-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the

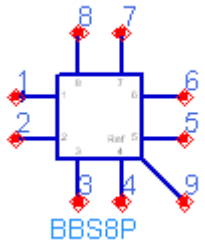
model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (8) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS8P (8-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 8-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the

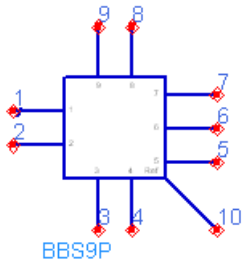
model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (9) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS9P (9-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 9-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to"Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.bbn (rational-polynomial formulation) and <*filename*>.*ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension .*bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (.*ckt*) model may provide better results.

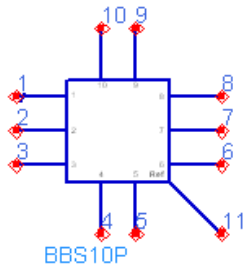
4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (10) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBS10P (10-Port Broadband SPICE Model)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an 1-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are <*filename*>.bbn (rational-polynomial formulation) and <*filename*>*.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt*) model may provide better results.

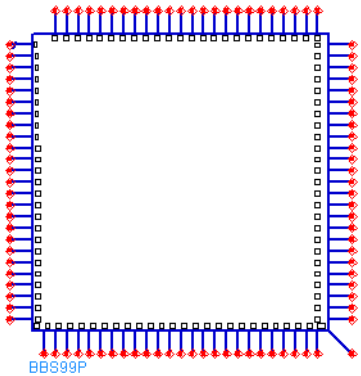
4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (11) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

BBSnP (Port Broadband SPICE Model n>99)

Symbol



Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current project directory	None	None

Notes and Equations

1 The BBSnP component is used to represent an n-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to "Using Broadband SPICE Models in ADS" on page 44.

2 Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).

3 In general, the rational polynomial model (with the extension *.bbn*) is recommended over the controlled

sources model. If the number of ports and/or the number of poles associated with the model is large, though, the controlled sources (*.ckt)* model may provide better results.

4 Broadband SPICE models are implemented as subcircuits, defined in hpeesofsim netlist fragments. The name of the model's top-level subcircuit is displayed in the BBSnP component's Edit Parameters dialog when a valid *ModelFile* is specified.

5 Ref pin (n+1) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

6 This component has no default artwork associated with it.

Index