

NOTICE: This document contains references to Agilent Technologies. Agilent's former Test and Measurement business has become Keysight Technologies. For more information, go to **[www.keysight.com](http://www.keysight.com)**.





**September 2011**  
**Broadband SPICE Model Generator**

© **Agilent Technologies, Inc. 2000-2011**

5301 Stevens Creek Blvd., Santa Clara, CA 95052 USA

No part of this documentation 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. Mentor products and processes are registered trademarks of Mentor Graphics Corporation. \* Calibre is a trademark of Mentor Graphics Corporation in the US and other countries. "Microsoft®, Windows®, MS Windows®, Windows NT®, Windows 2000® and Windows Internet Explorer® 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. Oracle and Java and registered trademarks of Oracle and/or its affiliates. Other names may be trademarks of their respective owners. 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. FLEXIm is a trademark of Globetrotter Software, Incorporated. Layout Boolean Engine by Klaas Holwerda, v1.7 <http://www.xs4all.nl/~kholwerd/bool.html> . FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg. QuestAgent search engine (c) 2000-2002, JObjects. Motif is a trademark of the Open Software Foundation. Netscape is a trademark of Netscape Communications Corporation. Netscape Portable Runtime (NSPR), Copyright (c) 1998-2003 The Mozilla Organization. A copy of the Mozilla Public License is at <http://www.mozilla.org/MPL/> . FFTW, The Fastest Fourier Transform in the West, Copyright (c) 1997-1999 Massachusetts Institute of Technology. All rights reserved.

The following third-party libraries are used by the NlogN Momentum solver:

"This program includes Metis 4.0, Copyright © 1998, Regents of the University of Minnesota", <http://www.cs.umn.edu/~metis> , METIS was written by George Karypis (karypis@cs.umn.edu).

Intel@ Math Kernel Library, <http://www.intel.com/software/products/mkl>

SuperLU\_MT version 2.0 - Copyright © 2003, The Regents of the University of California, through Lawrence Berkeley National Laboratory (subject to receipt of any required approvals from U.S. Dept. of Energy). All rights reserved. SuperLU Disclaimer: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS

INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

7-zip - 7-Zip Copyright: Copyright (C) 1999-2009 Igor Pavlov. Licenses for files are: 7z.dll: GNU LGPL + unRAR restriction, All other files: GNU LGPL. 7-zip License: This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA. unRAR copyright: The decompression engine for RAR archives was developed using source code of unRAR program. All copyrights to original unRAR code are owned by Alexander Roshal. unRAR License: The unRAR sources cannot be used to re-create the RAR compression algorithm, which is proprietary. Distribution of modified unRAR sources in separate form or as a part of other software is permitted, provided that it is clearly stated in the documentation and source comments that the code may not be used to develop a RAR (WinRAR) compatible archiver. 7-zip Availability: <http://www.7-zip.org/>

AMD Version 2.2 - AMD Notice: The AMD code was modified. Used by permission. AMD copyright: AMD Version 2.2, Copyright © 2007 by Timothy A. Davis, Patrick R. Amestoy, and Iain S. Duff. All Rights Reserved. AMD License: Your use or distribution of AMD or any modified version of AMD implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. AMD Availability: <http://www.cise.ufl.edu/research/sparse/amd>

UMFPACK 5.0.2 - UMFPACK Notice: The UMFPACK code was modified. Used by permission. UMFPACK Copyright: UMFPACK Copyright © 1995-2006 by Timothy A. Davis. All Rights Reserved. UMFPACK License: Your use or distribution of UMFPACK or any modified version of UMFPACK implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at

your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. UMFPACK Availability: <http://www.cise.ufl.edu/research/sparse/umfpack> UMFPACK (including versions 2.2.1 and earlier, in FORTRAN) is available at <http://www.cise.ufl.edu/research/sparse> . MA38 is available in the Harwell Subroutine Library. This version of UMFPACK includes a modified form of COLAMD Version 2.0, originally released on Jan. 31, 2000, also available at <http://www.cise.ufl.edu/research/sparse> . COLAMD V2.0 is also incorporated as a built-in function in MATLAB version 6.1, by The MathWorks, Inc. <http://www.mathworks.com> . COLAMD V1.0 appears as a column-preordering in SuperLU (SuperLU is available at <http://www.netlib.org> ). UMFPACK v4.0 is a built-in routine in MATLAB 6.5. UMFPACK v4.3 is a built-in routine in MATLAB 7.1.

Qt Version 4.6.3 - Qt Notice: The Qt code was modified. Used by permission. Qt copyright: Qt Version 4.6.3, Copyright (c) 2010 by Nokia Corporation. All Rights Reserved. Qt License: Your use or distribution of Qt or any modified version of Qt implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. Qt Availability: <http://www.qtsoftware.com/downloads> Patches Applied to Qt can be found in the installation at: \$HPEESOF\_DIR/prod/licenses/thirdparty/qt/patches. You may also contact Brian Buchanan at Agilent Inc. at [brian\\_buchanan@agilent.com](mailto:brian_buchanan@agilent.com) for more information.

The HiSIM\_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or 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 documentation 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** U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

About Broadband Spice Model Generator . . . . .	7
Creating Broadband SPICE Models . . . . .	23
Using Broadband Spice Models . . . . .	33
Passive Broadband SPICE Model Reference . . . . .	39

# About Broadband Spice Model Generator

The Broadband SPICE Model Generator tool converts the 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 the output model data in several different file formats, so that the data can be easily used with a number of different circuit simulators. For example, transient, frequency domain, envelope, and harmonic balance.

## Supported Operating Systems

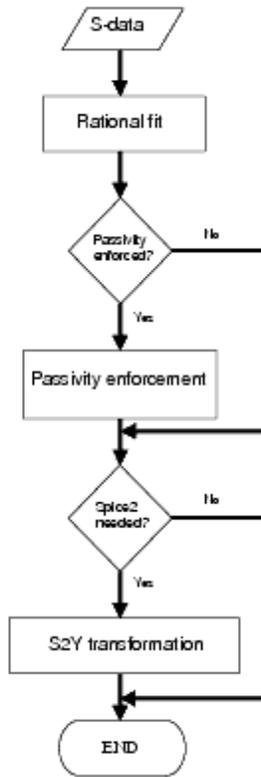
Broadband Spice Model Generator is supported on all operating systems supported by ADS. See the installation documentation for more details.

## Accessing the Documentation

To access the documentation for Broadband SPICE Model Generator, from any ADS window select **Help > Topics and Index > Simulation > Model Creation > 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:



## Using Broadband SPICE Model Generator

### 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, which is used to initialize the Broadband Spice tool 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 *<workspace>/<user>/<site>* basis like other ADS tools.

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

```

input.performSmoothing 0;
input.smoothingSize 1;
input.isMeasuredData 0;
model.dataSampling All;
model.targetAccuracy 5.000000e-3;
model.autoGenerate 1;
fitting.globalPolesImposed 1;
fitting.minDeltaPolesZeros -1;

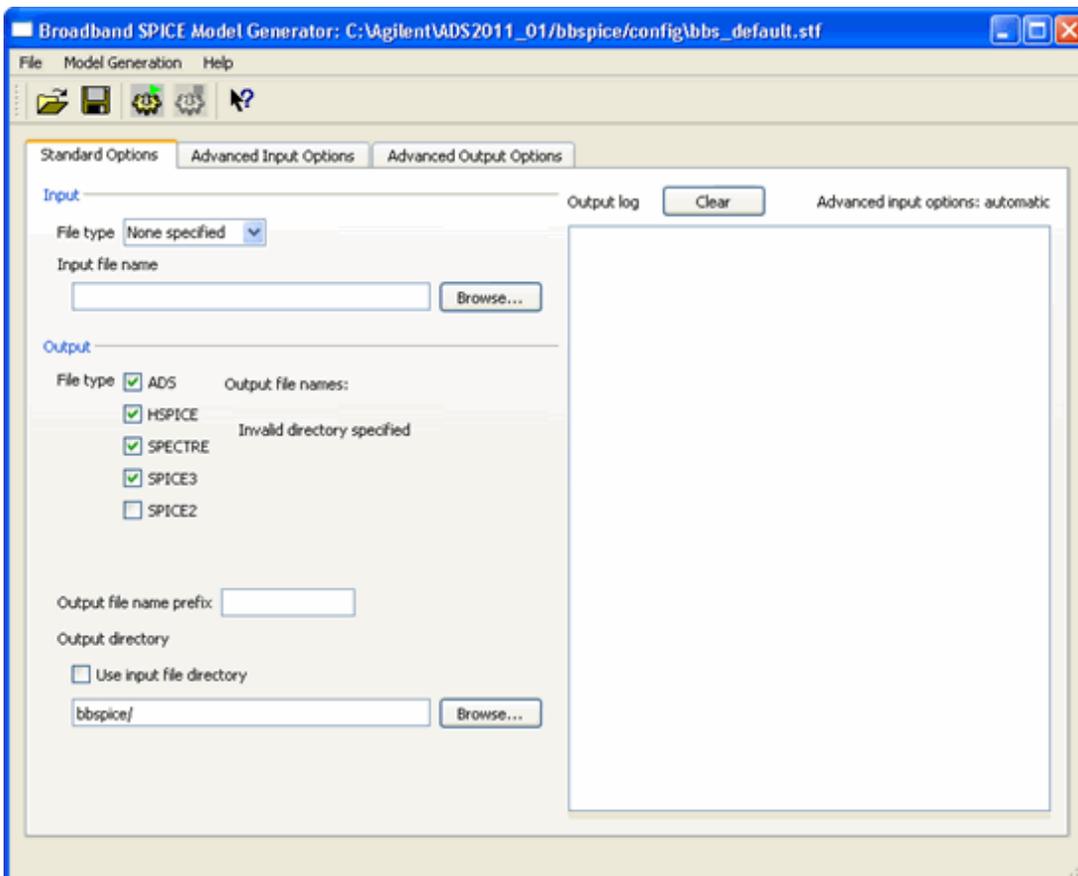
```

## Broadband SPICE Model Generator

```
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/;
dataset.opendds 1;
datadisplay.name S_Nport_BBSMG;
```

## Starting Broadband SPICE

To start the Broadband SPICE tool, select **Tools > Spice Model Generator > Start Broad Band Generator** from a Schematic window. The **Broadband Spice Model Generator** window is displayed, as shown below:



In the simplest operation, it is only necessary to specify the input file name 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.

The Broadband Spice Model Generator window provides three tabs: Standard, Advanced Input, and Advanced Output. These tabs are described in the following sections.

## Standard Options Tab

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

### Providing the Input Data

The input data used by Broadband Spice must contain S-parameter information with a reference impedance of 50  $\Omega$ .

**Note**  
If only other network parameters are available (such as, Y, Z, H, and ABCD), only S-data transformation is required.

### Selecting a File Type

The following figure displays the input files 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 workspace. 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.

**Note**  
The *.sam* file contains information about the frequency samples chosen by the Momentum AFS algorithm. It is recommended that you use the name of your layout as a prefix for the output model files.

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

**Default:** *None specified*

### 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.

### Specifying a File type

Two model files, *<filename>.ckt* and *<filename>.bbn*, are created when you select the **ADS** check box. These file types specifies two 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* (bbspice)

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

If **HSPICE** is selected, a *<filename>.hsp* is created. By default, a description using a pole/zero representation is used. An alternative representation using pole/residues can also be dumped. In order to do this, you must add a line " output.hspicePoleRes 1; " in the state file. This will generate a *<filename\_Foster>.hsp* file. Hspice versions from 2007 on will be able to process the pole/residue representation.

If **SPECTRE** is selected, a *<filename>.scs* file is created.

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.

**Caution**  
**Noise behavior of the models** Noise is suppressed in the generated models.

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

**Default:** *ADS* , *SPICE3* and *HSPICE*

### Output Log

This window displays status messages regarding various Broadband SPICE operations, for example, model generation.

### Output Log

Output log  Advanced input options: automatic

```

Model generation status:
-----
Beginning model generation.

Start rational model fitting
Rational model: convergence
RMS accuracy = 0 (fitting tolerance = 0.005)
Rational model : 2 poles

Start passivity enforcement
Passivity enforcement: successful
Output: ADS model file
Output: SPICE3 model file

The following output files were generated:
C:/users/default/PD513_BBSpiceUI_TEST_prj/data\SjTestbbsui_test.ds
C:/users/default/PD513_BBSpiceUI_TEST_prj/networks\stevenbbsui_test.bbn
C:/users/default/PD513_BBSpiceUI_TEST_prj/networks\stevenbbsui_test.ckt
C:/users/default/PD513_BBSpiceUI_TEST_prj/networks\stevenbbsui_test.sp3

Finished model generation.
-----
  
```

To empty the contents of 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*.

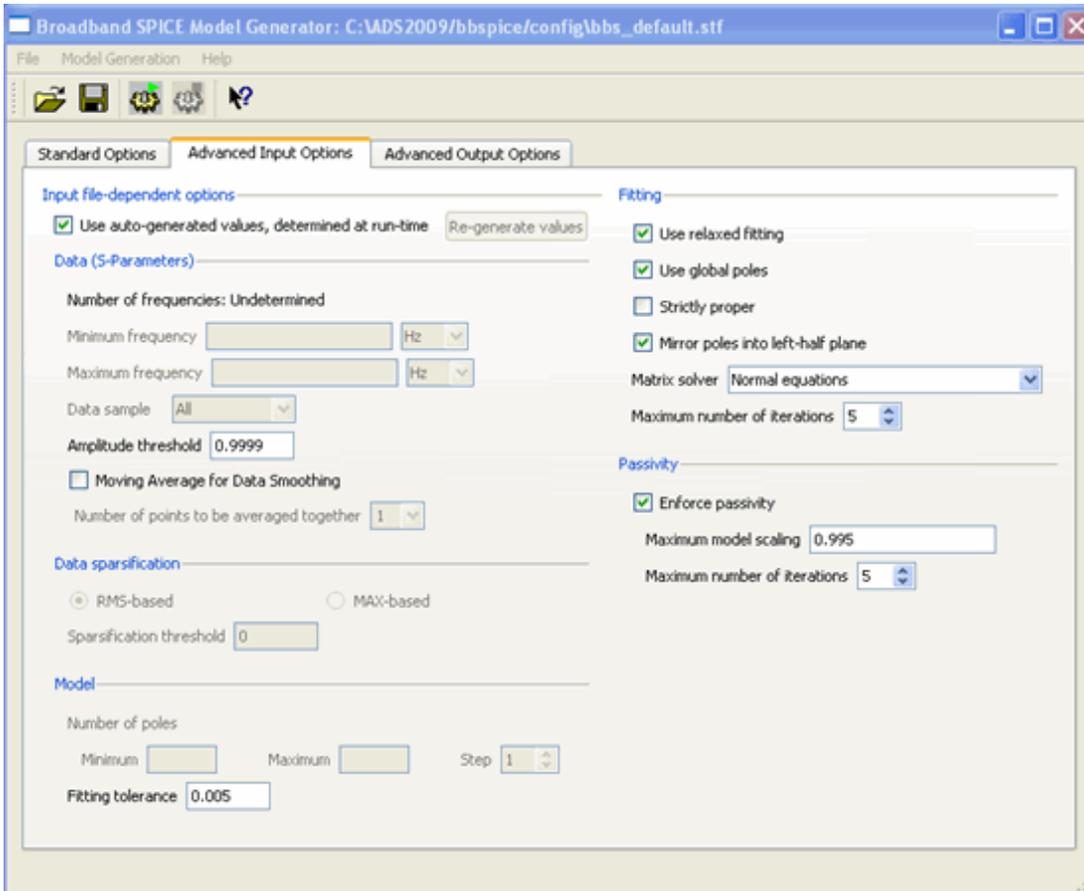
If *automatic*, the input data-dependent advanced option values are automatically calculated during model generation. If *specified*, the input data-dependent advanced option values specified in the dialog will be used.

## Advanced Input Options Tab

### Note

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

### Advanced Input Options Tab



The following selections 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.

## 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. A data file with too few frequency points will result in "Underdetermined set of equations" error. Lower the maximum number of poles in such case.

### 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

### Moving Average for Data Smoothing

This applies a moving average to the original input data which is used to smooth it out and prevent unnecessary noise from being modeled. As a rule, data containing unnecessary noise leads to a larger circuit size. This functionality is especially useful when

applied to measurement data, as it generally contains more noise than simulation data.

**Default:** *Un-checked*

### Number of Points to be Averaged Together

You can specify the number of points to be averaged together. This must be a positive odd number.

### Data sparsification

This dialog enables you to do a selection of S-parameters to be modeled dependent on their *RMS* or *maximum* value. By default, all S-parameters are selected. The sparsification threshold sets the Limit to decide whether a scattering parameter is taken into account. In the generated model the unselected S-parameters are set to zero, independent of the frequency. In order to see which input data is selected, test file *<filename>.sle* is generated.

**Default:**

RMS-based selected and Sparsification set to 0.

### 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:

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

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 relaxed fitting

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

**Default:** *Checked*

### 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:

$\mathbf{N} = \mathbf{D} - \mathbf{1}$ , with  $\mathbf{N}$  the polynomial degree of the numerator and  $\mathbf{D}$  the polynomial degree of the denominator of the rational model.

If the box is un-checked, the rational model is called *proper* and  $(\mathbf{D} - \mathbf{1}) \leq \mathbf{N} \leq \mathbf{D}$ .

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

**Default:** *Un-checked*

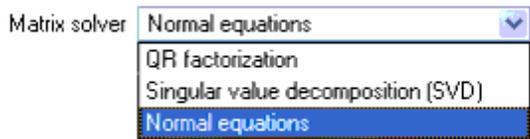
### Mirror poles into left-half plane

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 ( $\geq 1$ ). 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  $\geq 1$  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*

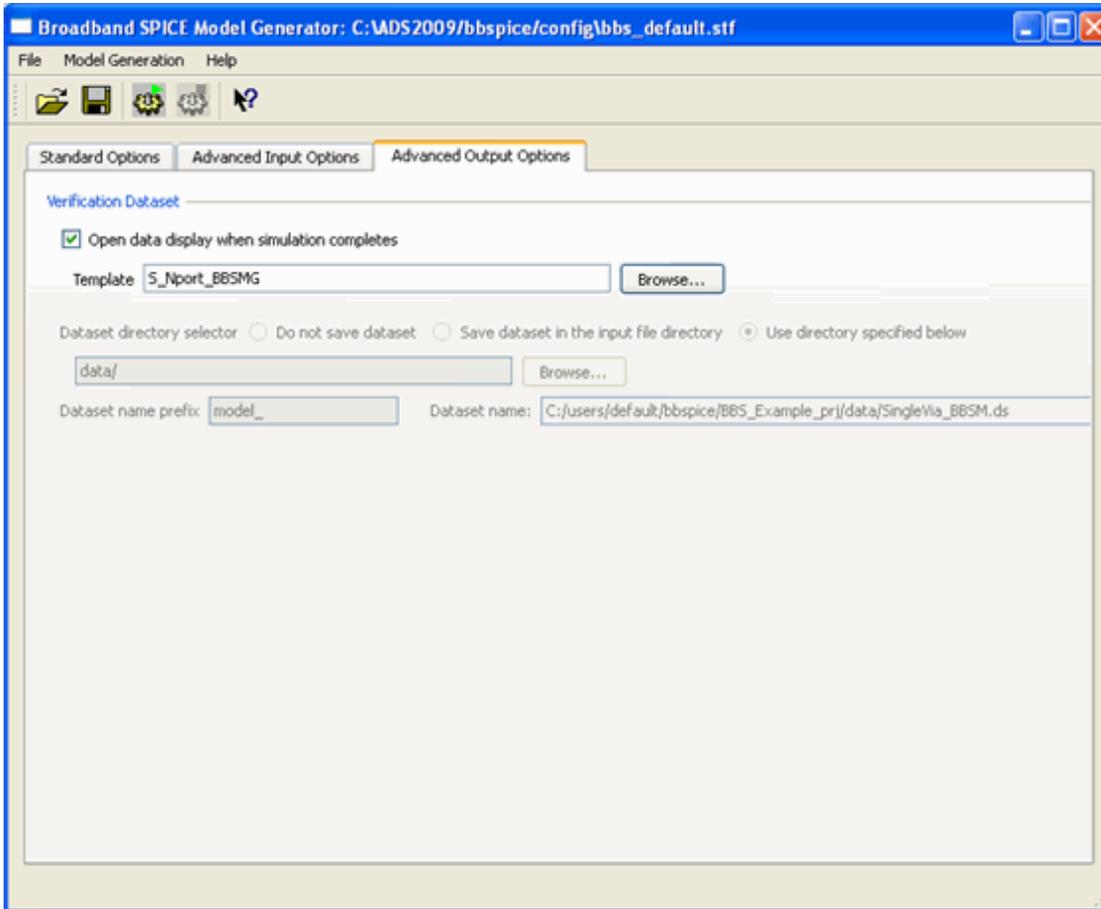
**Note**

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

### Advanced Output Options Tab

## 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.

The following selections are available under the *Advanced Output Options* tab.

### Verification Dataset

#### Open data display when simulation completes

This dialog enables you to automatically open a data display and a data display template after the model generation finished. Using this you can easily compare your original data to the modeled data in the display.

Checking this box enables this functionality.

**Default:** *Checked*

#### Template

This enables you to choose the template name that will be shown in the data display.

**Default:** `_S_Nport_BBSMG_`

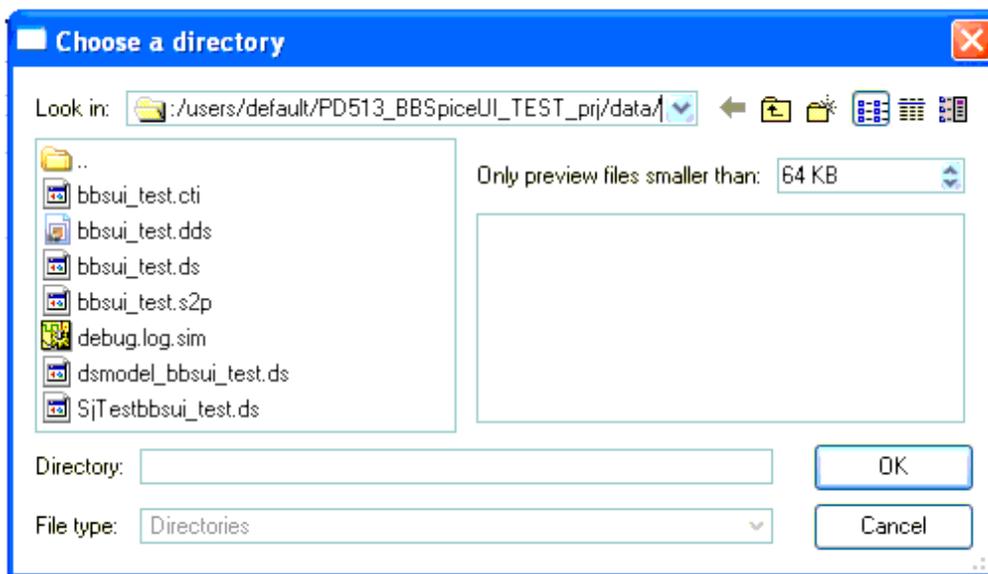
### Data directory selector

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.

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 workspaces data directory.



### Dataset name prefix

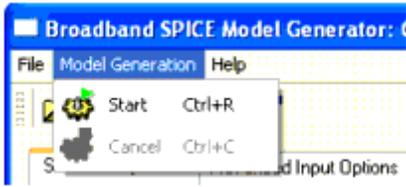
This enables you to choose a unique name for the output dataset.

### 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.



# Creating Broadband SPICE Models

This section 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 display 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). For using the broadband SPICE models in an ADS frequency-domain and time-domain circuit simulation, see *Using Broadband SPICE Models* (bbspice). The initial S-Parameter data is compared with the generated broadband SPICE models. It displays 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 ADS circuit simulators.

### 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.

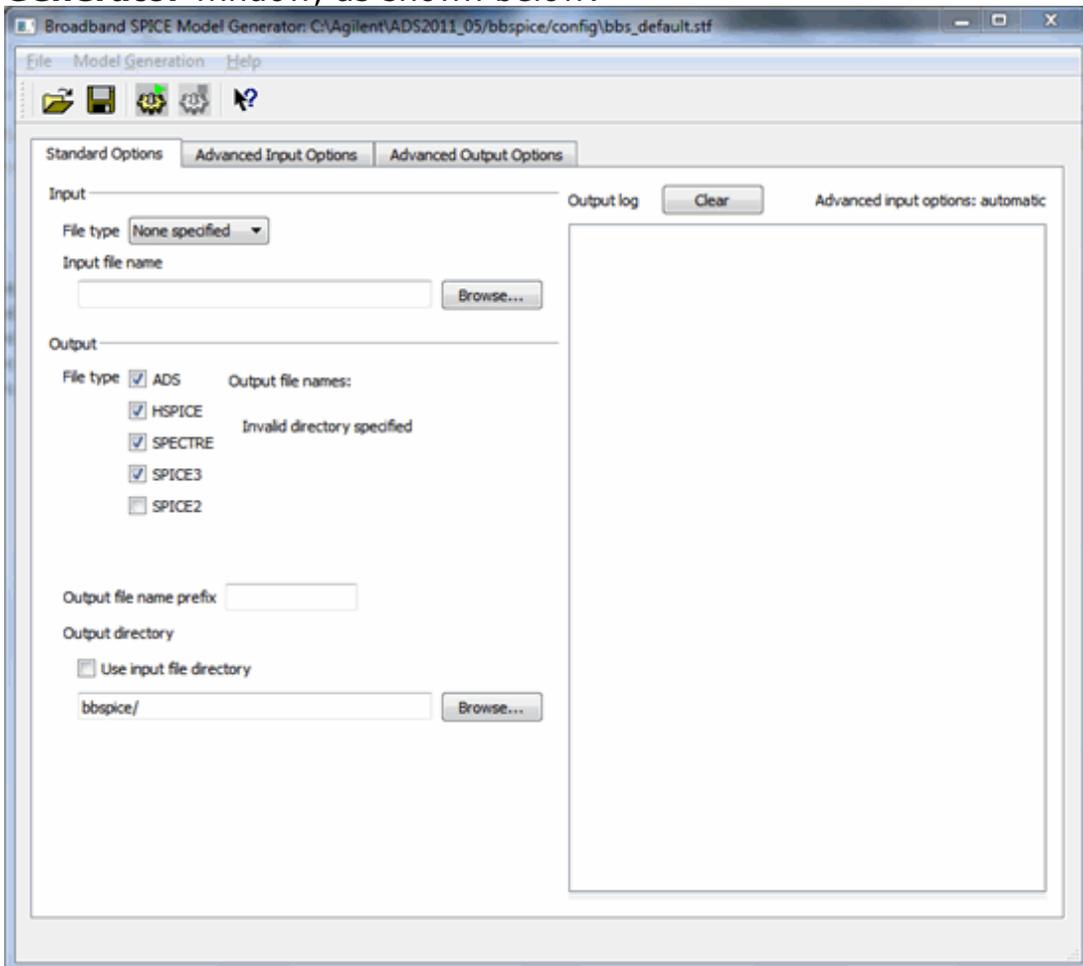
### Model Creation Using a Package Via Structure

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

The 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 Workspace or Project** and unarchive the required file.
3. Open a **Schematic** window.
4. From the **Schematic** window main menu bar, select **Tools > Spice Model Generator > Start Broad Band Generator** to open the **Broadband SPICE Model Generator** window, as shown below:



## Setting up the Broadband SPICE Model Generation Tool

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

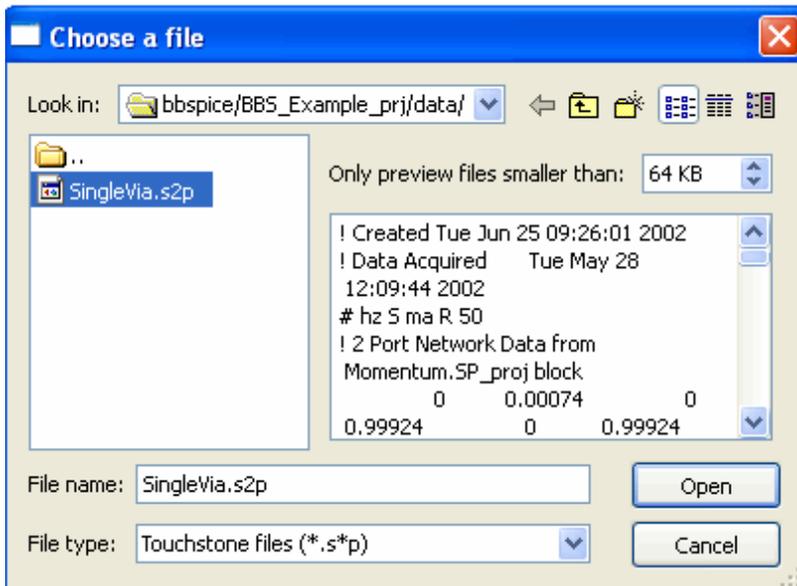
1. In the *Input* section of Broadband SPICE Model Generation Tool *Standard Options* tab, select the file from the *File type* drop-down list.



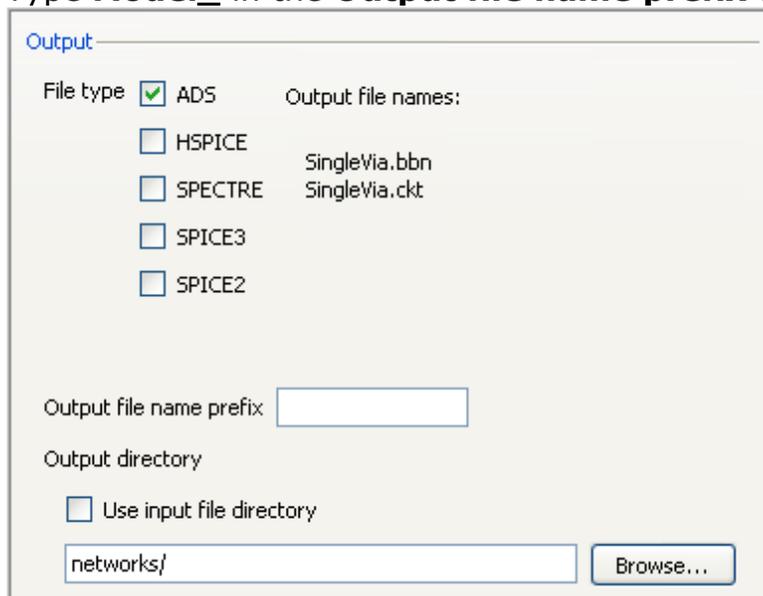
### Note

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

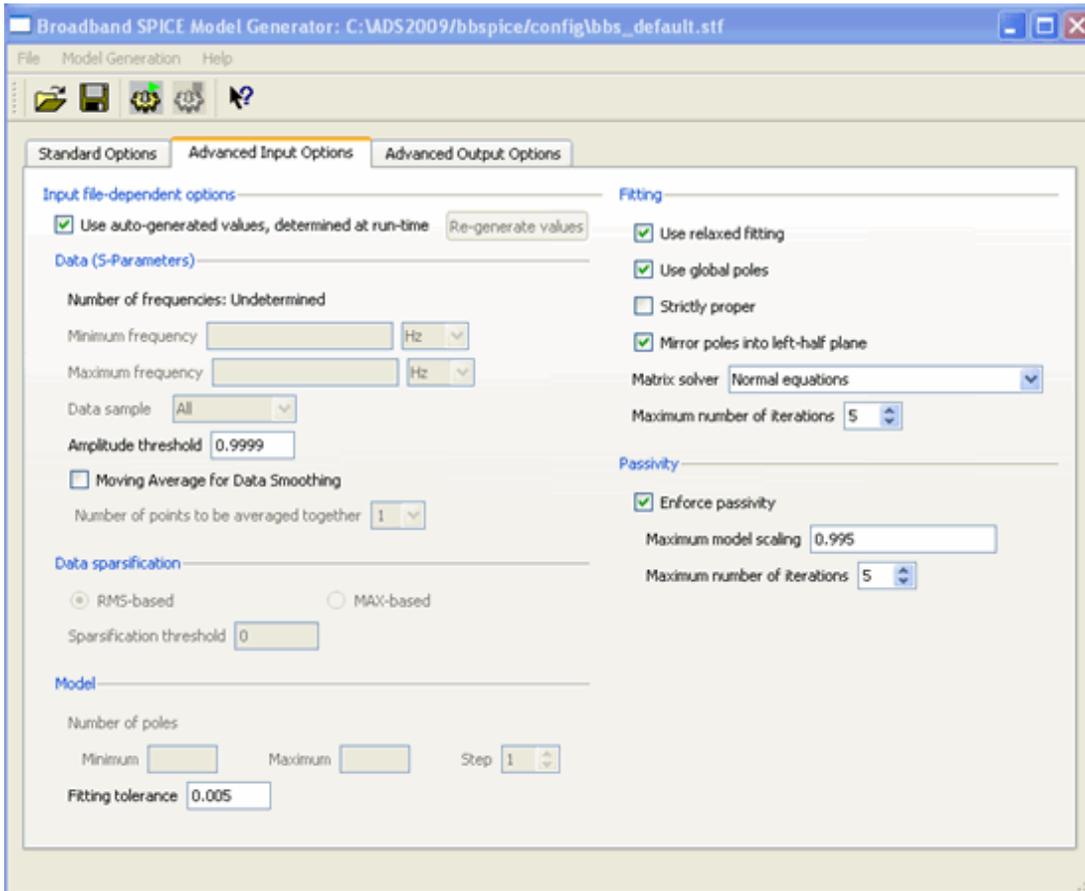
2. Click the **Browse** button associated with *Input File Name* dialog box and select the required file from the *Browse* window.



3. In the *Output* section, select **ADS** as the *Output File Type*.
4. Type **Model\_** in the **Output file name prefix** text box.



5. Select the **Advanced Input Options** tab.



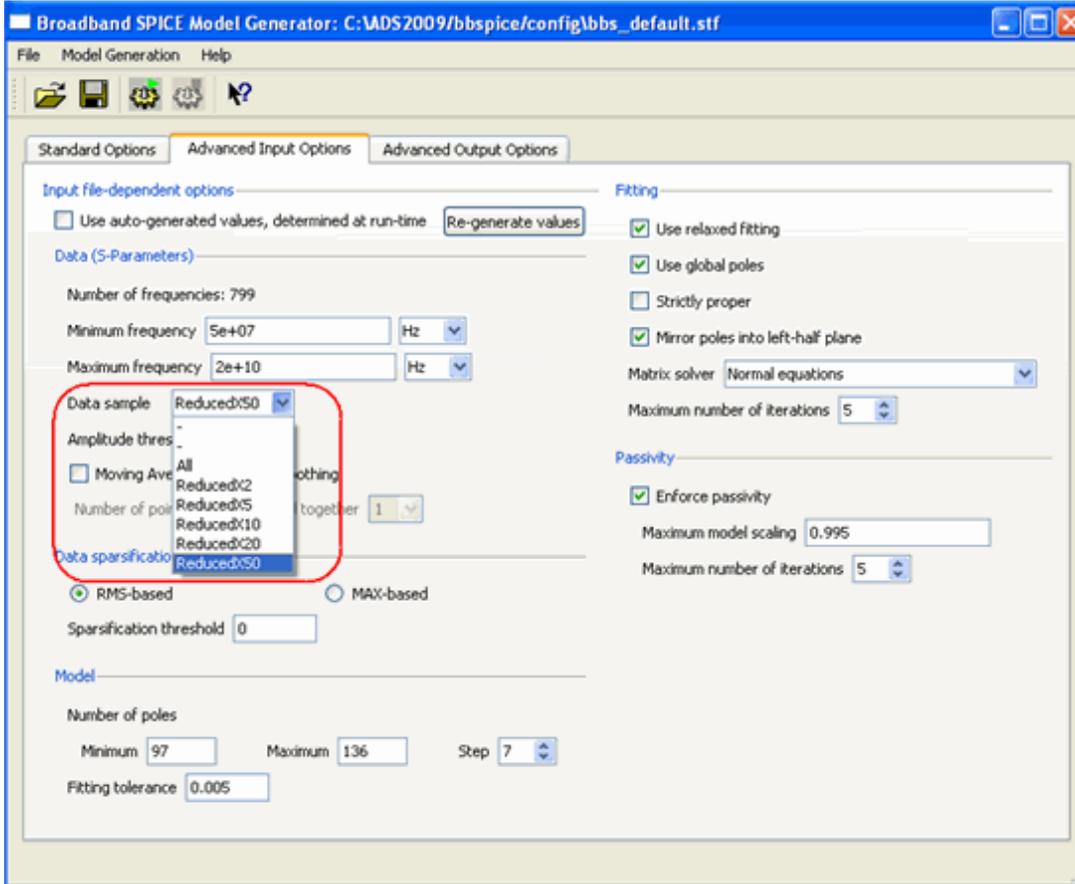
- Remove the selection from the **Use auto-generated values, determined at run-time** option. It generates the recommended initial values for the input data-dependent options, and activates the deactivated selections in the *Independent file-dependent* options section.



#### 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.

- In the *Data Sample* section of this window, select the required value from the drop-down list.



8. Select 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 box.

Output log  Advanced input options: automatic

```

Model generation status:
-----
Beginning model generation.
Start rational model fitting
Rational model: convergence
  RMS accuracy = 2.22077e-016 (fitting tolerance = 0.00500000)
  Rational model : 2 poles
Start passivity enforcement
The fitted model is passive.
Output: ADS model file
Output: HSPICE model file
Output: SPECTRE model file
Output: SPICE3 model file
The following output files were generated:
data/cell_3_BBSM.ds
bbspice/cell_3.bbn
bbspice/cell_3.hsp
bbspice/cell_3.scs
bbspice/cell_3.ckt
bbspice/cell_3.sp3

Finished model generation.
-----

```

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 box appears.
4. Exit **ADS**.

For information on how to use and verify these models in ADS, refer to *Using Broadband SPICE Models in ADS* (bbspice).

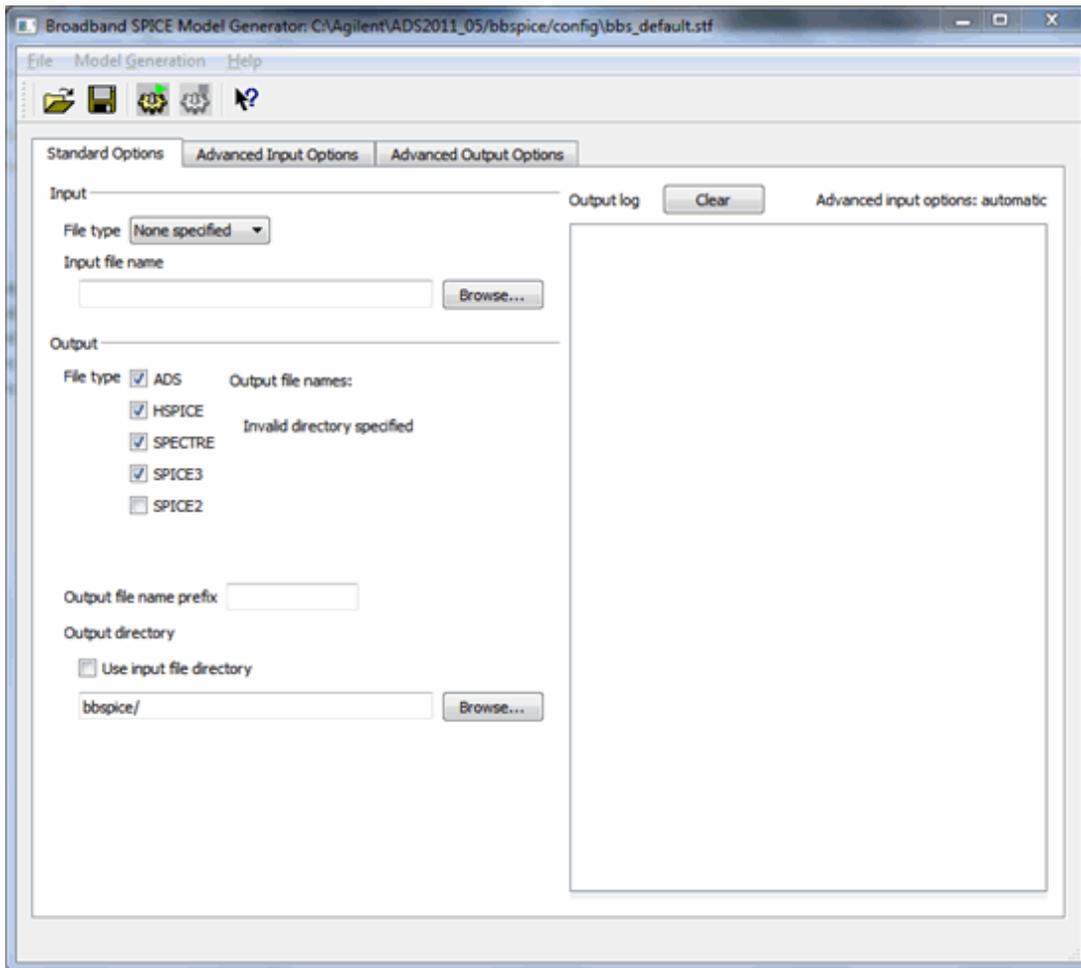
## Model Creation Using an FR4 Coupled-Microstrip Lines Circuit

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

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

### 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 the required file in the *File type* drop-down list.



### Note

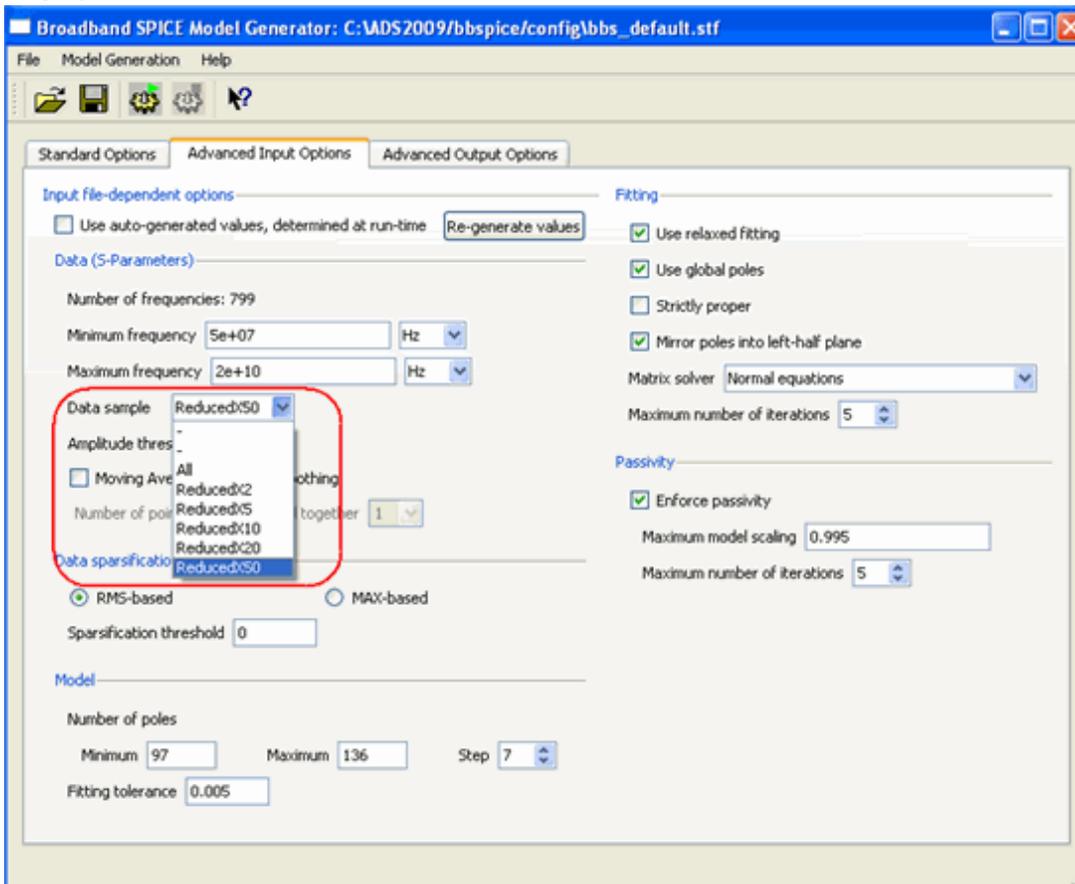
You must select a *File type* from in the *Input* section to access the *Advanced Input* and *Advanced Output* options tab.

2. Click the **Browse** button associated with *Input File Name* dialog and select the required file from the *Browse* window.
3. In the *Output* section, select **ADS** as the *Output File Type*.
4. Type " **Model\_** " in the *Output file name prefix* dialog box.
5. Click the **Advanced Input Options** tab.
6. Remove selection from the **Use auto-generated values, determined at run-time** option. It 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.

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



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

- 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.
- Check the *Output log*.

### Output Log

Output log  Advanced input options: automatic

```

Model generation status:
-----
Beginning model generation.
Start rational model fitting
Rational model: convergence
  RMS accuracy = 2.22077e-016 (fitting tolerance = 0.00500000)
  Rational model : 2 poles
Start passivity enforcement
The fitted model is passive.
Output: ADS model file
Output: HSPICE model file
Output: SPECTRE model file
Output: SPICE3 model file
The following output files were generated:
data/cell_3_BBSM.ds
bbsplice/cell_3.bbn
bbsplice/cell_3.hsp
bbsplice/cell_3.scs
bbsplice/cell_3.ckt
bbsplice/cell_3.sp3

Finished model generation.
-----

```

**Note**

If the accuracy of the rational model is insufficient, it is possible that selecting the Use Relaxed fitting option will improve the accuracy.

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.

3. This completes the Model creation. Select **File** > **Exit** to exit the Broadband SPICE Model Generation tool.
4. Click **No**, if the *Save Changes* dialog that appears.
5. Exit **ADS**.

For information on how to use and verify these models in ADS, refer to, *Using Broadband SPICE Models in ADS* (bbsplice).

## 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.

# Using Broadband Spice Models

This section explains how to use the models created with the Broadband Spice Model Generator in ADS.

## Verifying the Rational Fitting Model Data for the SingleVia Structure

To verify the Broadband SPICE rational model data:

1. Start **ADS**.
2. Open a *Schematic* window.
3. From the ADS Main window, choose **Window > New Data Display** to open a *Data Display* window.
4. From **Data Display**, select **Tools > Data File Tool**.
5. From the **dftool** main window, select the *Read data file into data set* option.
6. Select the required file from the *File format to read* and provide the dataset name.
7. Click the **Read File** button to finish the process.
8. Close the **dftool** window.
9. From the Data Display window, click the **Smith Chart** icon from the left toolbar to insert a Smith Chart plot in the Data Display window.
10. Insert the following traces on 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 appears in the *Data Display* window:

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.

11. Close the *Data Display* window.

## Verifying the Output Model for the SingleVia Structure

1. Start **ADS**.
2. Open a Schematic window, select **File > Open**.
3. Locate and open the design *SingleViaFreqdomain*. The following circuit should appear in the Schematic window.

### SingleViaFreqdomain

4. Double-click the SNP data item to open the *Edit Parameter* window.
5. Click **Browse** with the required file type.
6. Select the required file, click **Open**
7. Click **OK**.
8. Double-click the BBS2P component BBS\_BBN to open the *Edit Parameter* window.
9. Click **Browse** with the *ModelFile* parameter selected.
10. Select the file *Model\_SingleVia.bbn*, click **Open** in the *Select File* window.
11. Click **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.

12. Double-click the BBS2P component BBS\_CKT to bring up the *Edit Parameter* window.
13. Click the **Browse** button with the *ModelFile* parameter selected.
14. In the *Select File* window, set the filter to All Files (.). Then, select the *Model\_SingleVia.ckt* file.
15. 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 [SingleViaFreqdomain](#), 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**. Locate and open the design *SingleVia\_TimeDomain*.
2. The following circuit appears in the schematic window.

### Transient Simulation of a Broadband SPICE component

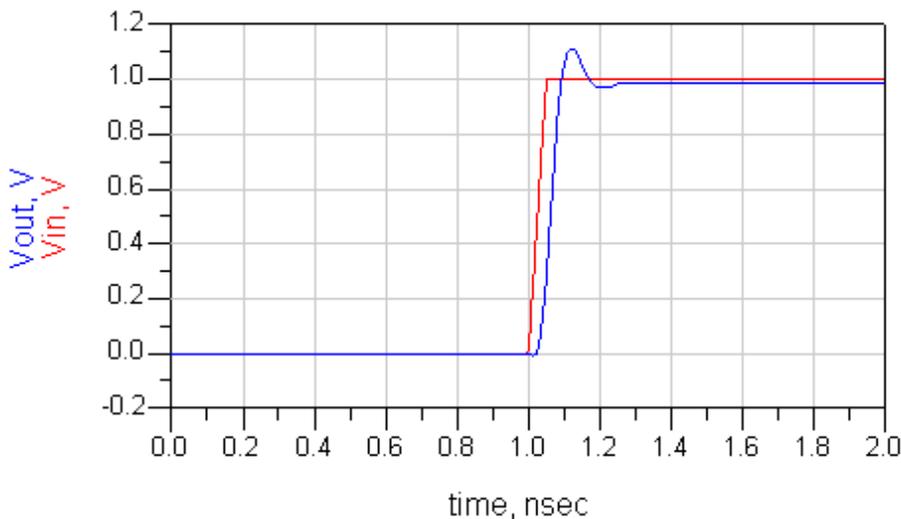
A transient simulation of the model represented by the BBS2P component has been set-up in this schematic. The following steps will ensure that the BBS2P component is set-up to correctly represent the Broadband SPICE model generated in *Creating Broadband SPICE Models* (bbsplice).

3. Double-click the **BBS2P component** to open the *Edit Parameter* window.
4. Click **Browse** button with the ModelFile parameter selected.
5. Select the file *Model\_SingleVia.bbn*, click **Open**.

**Note**

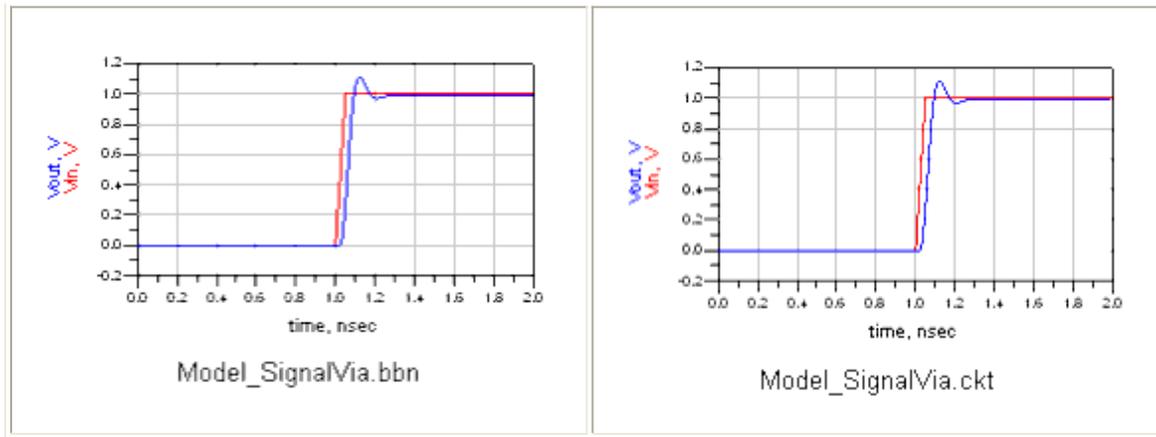
*Model\_SingleVia.bbn* is the ADS broadband SPICE output model file created and saved during the Broadband SPICE Model Generation.

6. Click **OK**.
7. Select **Simulate > Simulate** from the main Schematic window to perform the transient simulation.
8. When the simulation is complete, 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 appears in the Data Display window.

**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:



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

## Verifying 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](#) and [Verifying the Output Model for the SingleVia Structure](#).

## Using the CoupledMicrostrip Lines Model in the Time Domain

1. In a Schematic window, select **File > Open**. Locate and open the design *Coupledmicrostrip\_TimeDomain*. The following circuit appears in the schematic window.

### 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* (bbspice).

2. Double-click the **BBS4P component** to open the *Edit Parameter* window.
3. Click **Browse** 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* (bbspice)), you will use the *.ckt* file instead. Because of this, *All Files* need to be selected in order to be able to make the *.ckt* file selectable.
4. 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.

5. Click **OK**.
6. Select **Simulate** > **Simulate** from the main Schematic window to perform the transient simulation.
7. 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:

#### 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

While using the Broadband SPICE Model Generation tool, refer to the following guidelines

- If you have problem in obtaining the required convergence accuracy or passivity level, you should select the *Use relaxed fitting* option in the *Advanced Input Options* tab.
- If the number of poles are too high, the *.bbn* ADS model file will not be created, in most cases. In such cases, you should use the *.ckt* file instead of the *.bbn* file. However, if the *.bbn* file is available, you should 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*, it is recommended to select 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 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:

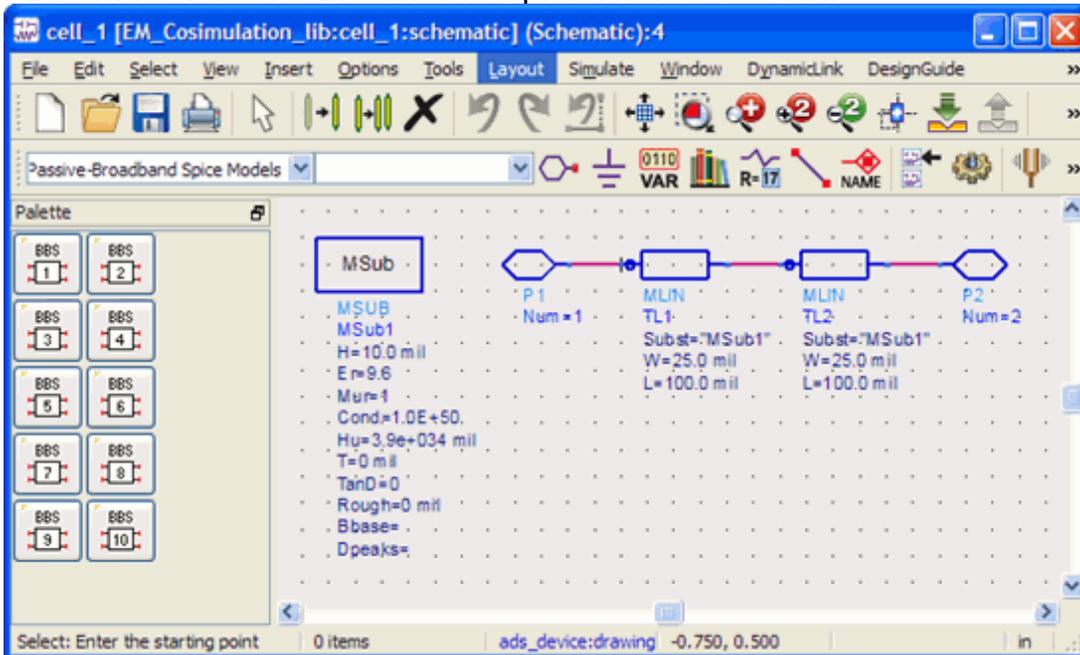
- If passivity enforcement is not needed and a passive rational fitted model is not required, it is best to uncheck the *Use global poles* option in the *Fitting* section of the *Advanced Input Options* tab.
- If passivity enforcement is not needed, but you want to know whether the fitted model is passive (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 and dynamical system with many ports (and thus a large simulation time), it is a good practice to run several simulations using all of the above settings (and any other combination, as you prefer). Then, save the output files with a different file name prefix to compare the results by using the *Data Display*.

# Passive Broadband SPICE Model Reference

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

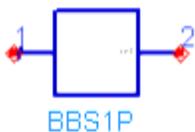
## Overview

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

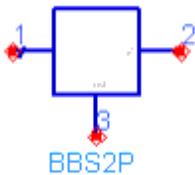
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model. The directory path is optional. The default directory is <prj>/ where <prj> is your current workspace directory	None	None

## Notes and Equations

- 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* (bbspice).
- Valid model file extensions are <filename> .bbn (rational-polynomial formulation) and <filename>.ckt (controlled sources formulation).
- 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.
- 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.
- Ref pin (2) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS2P (2-Port Broadband SPICE Model)

### Symbol



### Parameters

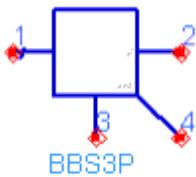
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

## Notes and Equations

- 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* (bb spice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin (3) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- 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.	None	None

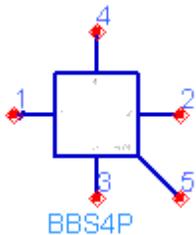
#### Notes and Equations

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

- 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.
- Ref pin (4) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS4P (4-Port Broadband SPICE Model)

### Symbol



### Parameters

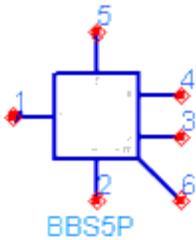
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

### Notes and Equations

- 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* (bbspice).
- Valid model file extensions are *<filename>.bn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- In general, the rational polynomial model (with the extension *.bn*) 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.
- 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.
- Ref pin (5) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS5P (5-Port Broadband SPICE Model)

## Symbol



## Parameters

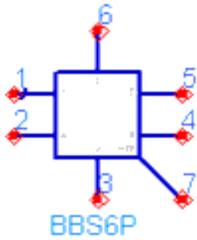
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

## Notes and Equations

- 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* (bbspice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin (6) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS6P (6-Port Broadband SPICE Model)

### Symbol



## Parameters

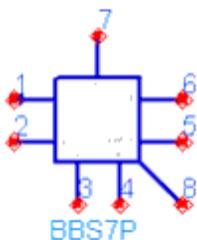
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

## Notes and Equations

- 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* (bbsplice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin (7) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS7P (7-Port Broadband SPICE Model)

### Symbol



## Parameters

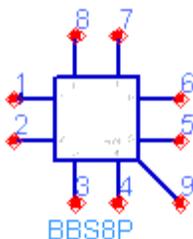
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

## Notes and Equations

- 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* (bbspice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin (8) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBS8P (8-Port Broadband SPICE Model)

### Symbol



## Parameters

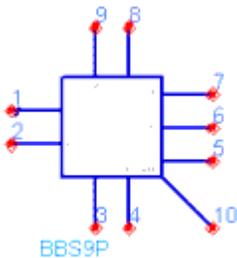
Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

## Notes and Equations

- 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* (bb spice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin (9) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- 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.	None	None

## Notes and Equations

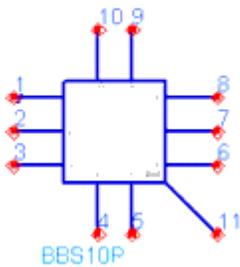
- The BBSnP component is used to represent an 9-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to *Using Broadband SPICE Models in ADS* (bb spice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation)

and *<filename>.ckt* (controlled sources formulation).

- In general, the rational polynomial model (with the extension *.bn*) 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.
- 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.
- Ref pin (10) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- 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.	None	None

### Notes and Equations

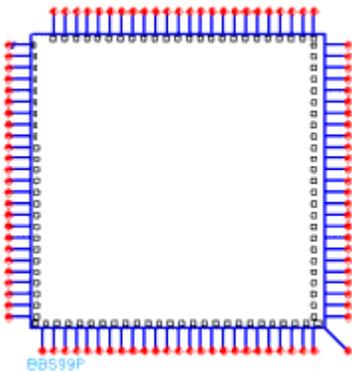
- The BBSnP component is used to represent an n-Port ADS model generated by the Broadband SPICE Model Generator tool. For information on Broadband SPICE model files, refer to *Using Broadband SPICE Models in ADS* (bbspice).
- Valid model file extensions are *<filename>.bn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- In general, the rational polynomial model (with the extension *.bn*) 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.
- 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.

- Ref pin (11) is the common terminal; it is normally grounded, but can be used in non-grounded mode.
- This component has no default artwork associated with it.

## BBSnP (Port Broadband SPICE Model $n \leq 99$ )

### Symbol



### Parameters

Name	Description	Units	Default
ModelFile	Name of data file containing n-port Broadband SPICE ADS model.	None	None

### Notes and Equations

- 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* (bbsplice).
- Valid model file extensions are *<filename>.bbn* (rational-polynomial formulation) and *<filename>.ckt* (controlled sources formulation).
- 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.
- 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.
- Ref pin ( $n+1$ ) is the common terminal; it is normally grounded, but can be used in non-grounded mode.

- This component has no default artwork associated with it.