



**Agilent Technologies**

# DC Simulation

**August 2005**

---

## Notice

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

Agilent Technologies makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

## Warranty

A copy of the specific warranty terms that apply to this software product is available upon request from your Agilent Technologies representative.

## Restricted Rights Legend

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

© Agilent Technologies, Inc. 1983-2005  
395 Page Mill Road, Palo Alto, CA 94304 U.S.A.

## Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries.

Microsoft<sup>®</sup>, Windows<sup>®</sup>, MS Windows<sup>®</sup>, Windows NT<sup>®</sup>, and MS-DOS<sup>®</sup> are U.S. registered trademarks of Microsoft Corporation.

Pentium<sup>®</sup> is a U.S. registered trademark of Intel Corporation.

PostScript<sup>®</sup> and Acrobat<sup>®</sup> are trademarks of Adobe Systems Incorporated.

UNIX<sup>®</sup> is a registered trademark of the Open Group.

Java<sup>™</sup> is a U.S. trademark of Sun Microsystems, Inc.

SystemC<sup>®</sup> is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission.

# Contents

## 1 DC Simulation

Overview.....	1-1
Using DC Simulation .....	1-3
License Requirements.....	1-3
When to Use DC Simulation.....	1-3
How to Use DC Simulation.....	1-4
What Happens During DC Simulation.....	1-4
Examples in ADS.....	1-5
Simulating a BJT in ADS.....	1-5
Sweeping Parameters in ADS.....	1-7
Example in RFDE.....	1-11
Sweeping Parameters in RFDE.....	1-11
DC Simulation Parameters .....	1-14
Saving Annotation Data in RFDE.....	1-14
Setting up a Sweep in ADS.....	1-16
Setting up a Sweep in RFDE.....	1-18
About Preparing Sweeps.....	1-20
Defining Simulation Parameters.....	1-21
Additional Parameters.....	1-26
Theory of Operation .....	1-27
Simulation Basics.....	1-27
Simulation Assumptions.....	1-27
For More Details.....	1-28
Troubleshooting a Simulation.....	1-29
Common Setup Errors.....	1-30
Adjusting Tolerance Parameters.....	1-31
Topology Check and Correction.....	1-32
Impossible Circuits.....	1-36
Model Parameter I <sub>max</sub> is Too Small.....	1-36
References .....	1-36

## Index



# Chapter 1: DC Simulation

This documentation describes DC simulation used in ADS and RFDE. It outlines when to use a DC simulation, how to set it up, and the data it generates. Examples are provided to show how to use this simulation. Detailed information describes the parameters, theory of operation, and troubleshooting information.

## Overview

DC simulation calculates the DC operating point characteristics of a design under test (DUT). DC analysis is fundamental to all Analog/RF simulations, so it is used for all Analog/RF designs. Following a topology check, the simulator performs an analysis of the DC operating point, including the circuit's power consumption.

You can also set up the DC simulation to sweep one or more parameters, enabling you to perform tasks such as verifying model parameters by comparing the simulated DC transfer characteristics (I-V curves) of the model with actual measurements.

In ADS, the DC simulation component is available in the Simulation-DC palette.

In RFDE, the DC simulation is one of the analysis choices for the ADSSim simulator.

Refer to the following topics for details on DC simulation:

- [“Using DC Simulation” on page 1-3](#) explains when to use DC simulation, describes the minimum setup requirements, and gives a brief explanation of the DC simulation process.
- [“Examples in ADS” on page 1-5](#) contains detailed examples of DC simulation setups for ADS, one for calculating a single DC point and another that sweeps variables to generate a set of I-V curves.
- [“Example in RFDE” on page 1-11](#) contains a detailed example of a swept DC simulation setup for RFDE.
- [“DC Simulation Parameters” on page 1-14](#) provides details about the parameters available in the DC simulation setup dialog.

- **“Theory of Operation” on page 1-27** provides details about DC simulation operation. This is provided if you have an interest in understanding the underlying technology. Typical users should not need to examine this material.
- **“Troubleshooting a Simulation” on page 1-29** offers suggestions on what to do in the rare event that a DC simulation does not converge or converges too slowly.
- **“References” on page 1-36** lists the references relevant to the theory of operation. This is provided if you have an interest in understanding the underlying technology. Typical users should not need to examine this material.

# Using DC Simulation

This section describes when to use a DC simulation, how to set it up, and the basic simulation process used to collect data.

## License Requirements

The DC simulation uses the Linear Simulator license (`sim_linear`). You must have this license to run DC simulations. You can work with examples described here and installed with the software without the license, but you will not be able to simulate them.

## When to Use DC Simulation

A single-point DC simulation automatically precedes every AC, S-parameter, transient, harmonic balance, and circuit envelope simulation. This serves as the starting point for these simulations. For AC and S-parameter simulations, it determines linearized models for the nonlinear components. For transient, harmonic balance, and circuit envelope simulations, it determines an initial estimate used for nonlinear simulation.

A DC simulation also can be done manually in ADS by using the DC simulation controller, and in RFDE by selecting the DC analysis. Start by creating your design, add any relevant current probes, then identify and name the nodes from which you want to collect data. The DC simulation setup then provides for both single-point and swept simulations. Swept variables can be related to voltage or current source values, or to other component parameter values. By performing a DC swept bias or swept variable simulation, you can check the operating point of the circuit against a swept parameter such as a bias supply voltage or a temperature.

Use DC simulation to:

- Verify the proper DC operating characteristics of the design under test.
- Determine the power consumption of your circuit.
- Verify model parameters by comparing the DC transfer characteristics (I-V curves) of the model with actual measurements.
- Display voltages and currents after a simulation.
- Provide data for DC back-annotation.

## How to Use DC Simulation

The following guidelines will help you set up a DC simulation.

In ADS:

- Add the DC simulation component to the schematic. If you do not edit the default settings, values will be calculated for the current probes and nodes added to the circuit, based on the settings of the schematic's other components.
- To sweep a parameter over a range, such as varying an input voltage or changing a resistor value, double-click the simulation component and select the Sweep tab. Enter the name of the parameter to sweep. Select the sweep type and enter the range.
- If additional sweeps are needed, an external parameter sweep can be added.
- For details about each parameter, click *Help* from the open dialog box.

In RFDE:

- Select the DC analysis from the Choosing Analyses window. If you do not edit the default settings, values will be calculated for the current probes and nodes that you have applied to the circuit, based on the settings of the other components in the schematic.
- To sweep a parameter over a range, such as varying an input voltage or changing a resistor value, adjust the parameters in the Parameter Sweep section of the Choosing Analyses window. Select the *Parameter Type*, and enter the parameter's *Variable Name* to sweep. Select the sweep type and enter the range.
- If additional sweeps are needed, an external parameter sweep can be added.
- For additional parameter settings, click *Options*.
- For details about each parameter, click *Help* from the open dialog box.

## What Happens During DC Simulation

The DC simulation technique relies on an iterative process of mathematical convergence toward a solution. By manipulating a system of nonlinear ordinary differential equations, it solves for an equilibrium point in the nonlinear algebraic equations that describe a circuit based on a set of assumptions. For details, see [“Theory of Operation” on page 1-27](#).

# Examples in ADS

There are two examples that describe how to set up and run a DC simulation in ADS:

- “[Simulating a BJT in ADS](#)” on page 1-5 shows how to calculate a single-point DC bias of a BJT.
- “[Sweeping Parameters in ADS](#)” on page 1-7 shows how to sweep one or more circuit parameters over a range of values during a simulation.

## Simulating a BJT in ADS

[Figure 1-1](#) illustrates the setup for a DC simulation of a BJT. This simulation calculates a single DC operating point.

---

**Note** This design, *DC1.dsn*, is in the *Examples* directory under *Tutorial/SimModels\_prj*. The results are in *DC1.dds*.

---

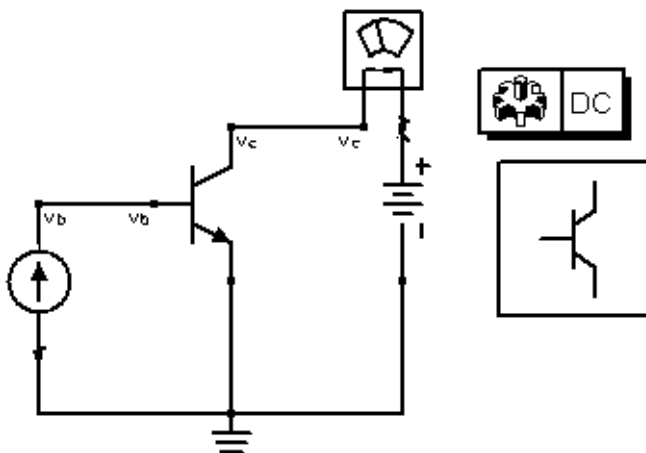


Figure 1-1. Setup for a single-point DC bias simulation

To perform a single-point DC simulation:

1. Place the BJT instance and model on the schematic.
2. From the **Sources-Freq Domain** palette, select **V\_DC**. Place the DC voltage source on the schematic to provide the collector voltage and modify the voltage as needed.

---

**Note** Ensure that sources are connected either by wires or by means of a named connection (*Insert > Wire/Pin Label*).

---

3. From the **Sources-Freq Domain** palette, select **I\_DC**. Place the source on the schematic to provide base current and modify the current as needed.
4. From the **Probe Components** palette, select **I\_Probe**. Place this current probe in an appropriate location in the circuit.
5. From the **Simulation-DC** palette, select and place the **DC** simulation component on the schematic.
6. **Simulate**. When the simulation is finished, you can immediately exercise the following DC simulation options:
  - For a direct representation of currents and voltages at all nodes on the schematic, choose *Simulate > Annotate DC Solution*.
  - For the details of the device, choose *Simulate > Detailed Device Operating Point* or *Brief Device Operating Point*, then click the device when the crosshairs appear.
  - Choose *Simulate > Clear DC Annotation* to remove the annotation.
7. In the Data Display, the data from the simulation is presented as a list of DC currents and voltages:

freq	DC1..Vc	DC1..Vb	..._Probe1.i
0.0000 Hz	3.000 V	678.8mV	61.02uA

## Sweeping Parameters in ADS

By performing a DC swept bias or a swept variable simulation, you can check the operating point of the circuit against a swept parameter such as temperature or bias supply voltage. The resulting data can be plotted to resemble the appearance of a curve tracer.

Figure 1-2 illustrates a setup for performing an idealized, swept DC bias simulation of a MOSFET. Vdd and Vgg are swept across a range of voltages. The result is a family of curves representing drain current versus drain voltage, for varying values of gate voltage.

---

**Note** This design, *DC2.dsn*, is in the *Examples* directory under *Tutorial/SimModels\_prj*. The results are in *DC2.dds*.

---

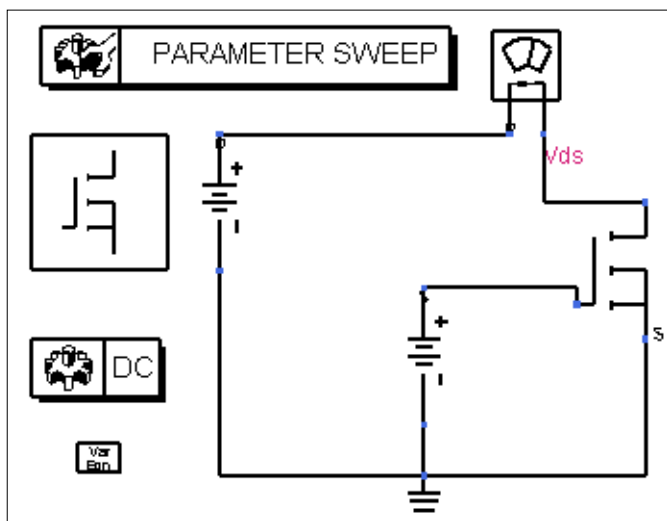


Figure 1-2. Example setup for a swept DC simulation

---

**Note** This example uses a referenced model. The parameter *cells* refers to the extraction parameter known as *binning* and reflects the electrical characteristics of this particular model. The number of cells has been referenced in a VarEqn component.

---

To perform an idealized, swept DC bias simulation:

1. Place the MOSFET instance and model on the schematic.
2. From the **Sources-Freq Domain** palette, select **V\_DC**. Place this V\_DC component (SRC1) between the base of the device and ground. Edit this component and define  $V_{dc} = V_{gg1}$ , rather than a numerical value.
3. Place another V\_DC component (SRC2) between the drain of the device and ground. Open this component and define  $V_{dc} = V_{dd1}$ , rather than a numerical value.

This makes it possible to sweep these values. Any swept parameter must be initialized by means of an equation, as will be demonstrated later.

4. From the **Probe Components** palette, select **I\_Probe**. Place the probe between the drain and SRC2, and rename it if you like (in this case, it has been called Probe1).
5. From the **Simulation-DC** palette, select and place the **DC** simulation component on the schematic.

6. Edit the **DC** component by selecting the **Sweep** tab and modifying the sweep parameters as follows:

- Parameter to sweep = **Vdd1**. This appears as *SweepVar* on the schematic if this parameter is displayed on the schematic.

---

**Note** Variables entered in this field will appear in quotes on the schematic. If you enter a variable directly on the schematic (in this case, as the right-hand side of the SweepVar statement of the DC component), you must surround the variable with double quotes. This applies to the other user-defined variables shown in subsequent steps.

---

- Start = **0**
- Stop = **6**
- Step = **0.1**

7. Click **OK** to accept changes and close the dialog box.

8. Choose **Simulation-DC** palette, select **Prm Swp** (ParamSweep). Place this component on the schematic and edit it as follows:

On the Sweep tab:

- Parameter to sweep = **Vgg1**
- Start = **2**
- Stop = **3**
- Step = **0.25**

On the Simulations tab:

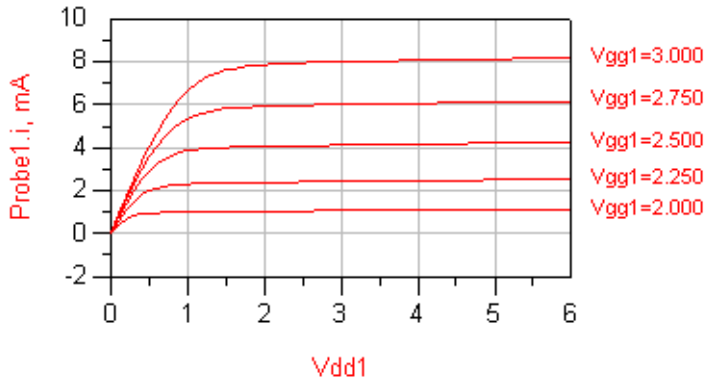
- Simulation 1 = **DC1**. This appears as *SimInstanceName[1]* on the schematic if this parameter is displayed on the schematic.

9. Click **OK** to accept changes and close the dialog box.

10. Choose the **Data Items** palette, select **Var eqn** (Variables and equations). Place this component on the schematic and enter the following equations:

- **Vdd1 = 4**
- **Vgg1 = 2.5**

11. Click **OK** to accept changes and close the dialog box.
12. **Simulate**. When the simulation is finished, plot the probe current (Probe1.i). The resulting family of curves representing drain current versus drain voltage, for varying values of gate voltage, is shown next:



# Example in RFDE

The following example describes how to set up and run a DC simulation in RFDE:

“Sweeping Parameters in RFDE” on page 1-11 shows how to sweep one or more circuit parameters over a range of values during a simulation.

## Sweeping Parameters in RFDE

Figure 1-3 illustrates the setup for simulation of amplifier DC transfer characteristics.

---

**Note** This cell, *Amp\_DC\_Swp*, is in the library *RFIC\_BJT\_Amp* under *CIW > Tools > RFDE Examples*.

---

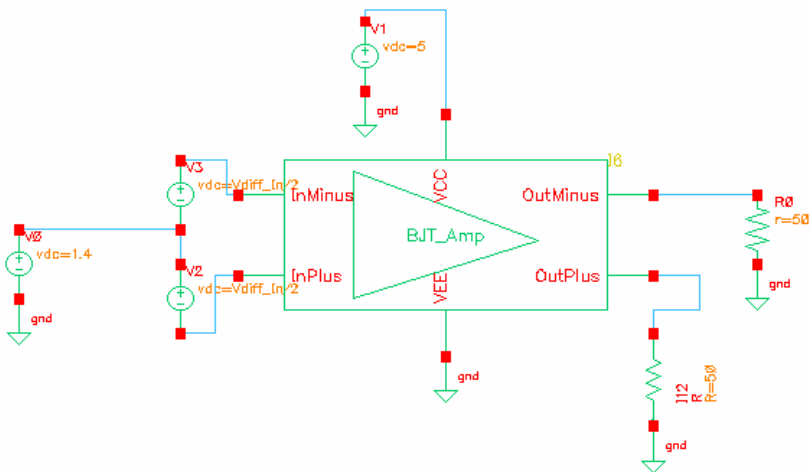


Figure 1-3. Setup for simulation of amplifier DC transfer characteristics

To perform the DC simulation:

1. Load the example *RFIC\_BJT\_Amp* using **CIW > Tools > RFDE Examples > Load Examples**.
2. Open the cell *Amp\_DC\_Swp* using **CIW > Tools > RFDE Examples > Open Examples**.
3. **Simulate**. When the simulation is finished, the following results should appear in the Data Display as shown in [Figure 1-4](#):

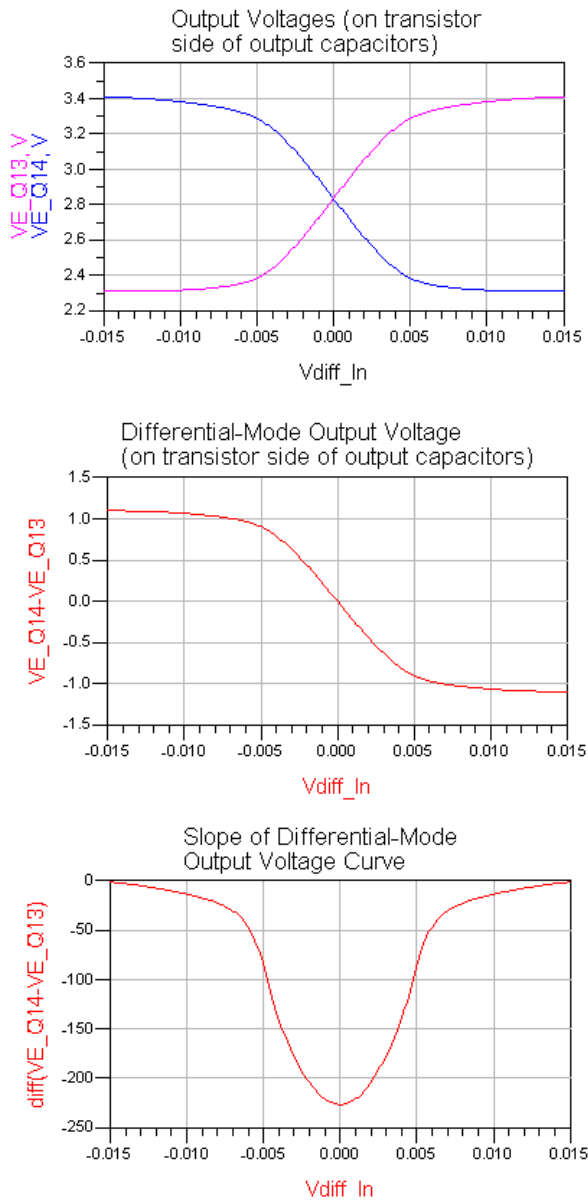


Figure 1-4. Data from simulation of amplifier DC transfer characteristics

## DC Simulation Parameters

The DC simulation setup dialog enables you to define the following aspects of the simulation:

- **Annotation Data (in RFDE)**—Saving DC data for schematic annotation.
- **Sweep**—Defining sweep characteristics.
- **Parameters**—Setting status level for summary information. Provides an option to output solutions at all steps, and access to advanced parameters related to convergence. Additionally:  
In ADS, setting device operating-point levels.  
In RFDE, setting sensitivities.
- **Output**—  
In ADS, selectively save simulation data to a dataset. For details, refer to the topic “Selectively Saving and Controlling Simulation Data” in the chapter “Simulation Basics” in the *Using Circuit Simulators* documentation.  
In RFDE, use *Outputs > Save Options* in the Analog Design Environment window.
- **Display (in ADS)**—Control the visibility of simulation parameters on the schematic. For details, refer to the topic “Displaying Simulation Parameters on the Schematic” in the chapter “Simulation Basics” in the *Using Circuit Simulators* documentation.

### Saving Annotation Data in RFDE

DC analysis in RFDE can output DC data for annotation to the schematic. The output (saved in a PSF file) contains node voltages, device currents, and device operating point data.

To save data, choose DC analysis in the Choosing Analysis dialog, click *Save Voltages and Currents* to save voltages and currents, and set *Device Operating Point Level* to *Brief* or *Detailed*. Note that *Save Voltages and Currents* must be checked in order to annotate device operating point information onto a schematic, it is not enough to set *Device Operating Point Level* to *Brief* or *Detailed*.

After running a simulation, you can annotate the schematic with the saved data. In the Analog Design Environment window, click *Results > Annotate* and select the desired option(s).

Sometimes, components from the *analogLib* library may not display operating point data. This happens when there is no ADSSim equivalent of Spectre operating point parameters. To display the missing annotations automatically, you must update the component instances in the Schematic window:

1. Select **Edit > Component Display**, then select the particular *analogLib* component.
2. In the Edit Component form, set *Select Label* to **parameter**.
3. In the *Parameter Labels* area, choose *Apply To library*, and **operating point**.
4. For the parameter names set to **none**, select parameters you want to display, and click **OK**.

In the following table, names used in netlists appear under *Parameter Name*.

**Table 1-1. RFDE Annotation Data Parameters**

Setup Dialog Name	Parameter Name	Description
Save Voltages and Currents	SaveDCVoltageSolution SaveDCCurrentSolution	Saves node voltages device currents, and optionally, device operating point information for annotation to the schematic. Default setting is disabled.
Device operating point level	DevOpPtLevel	Options to save device operating point information for most active devices and some linear devices in the circuit to the dataset. Default setting is None.
None	None (0)	No information is saved.
Brief	Brief (2)	Saves device currents, power, and some linearized device parameters.
Detailed	Detailed (4)	Saves the operating point values which include the device's currents, power, voltages, and linearized device parameters.

## Setting up a Sweep in ADS

Setting up the sweep portion of the simulation consists of three basic parts:

- Identifying the parameter you want to sweep.
- Selecting the sweep type and setting the associated characteristics.
- Optionally, specifying a sweep plan.

To shorten simulation time in any parameter sweep, select a start point where convergence is easy, and vary the parameter gradually. When selecting a start point, if you find that convergence is easier at one end of a sweep and harder at the other, use the easy end as the start of the sweep. Varying the parameter gradually yields better estimates for the next simulation, and achieves convergence more rapidly than if the parameter is changed abruptly. However, too small a step can lead to a prohibitively large number of sweep points, so be careful. For additional information about parameter sweep techniques, see [“About Preparing Sweeps” on page 1-20](#).

In the following table, names used in netlists and ADS schematics appear under *Parameter Name*.

Table 1-2. ADS DC Simulation Sweep Parameters

Setup Dialog Name	Parameter Name	Description
Parameter to sweep	SweepVar	The name of any defined parameter to be swept.
Parameter sweep—The sweep type and parameters. Disable <i>Use sweep plan</i> to set these parameters.		
Sweep Type		
Single point	Pt	Enables simulation at a specific value for the parameter. Specify the desired value in the <i>Parameter</i> field.
Linear		Enables sweeping a range of values based on a linear increment. Click <i>Start/Stop</i> to set start and stop values for the sweep, or <i>Center/Span</i> to set the center value and a span of the sweep.
Log		Enables sweeping a range of values based on a logarithmic increment. Click <i>Start/Stop</i> to set start and stop values for the sweep, or <i>Center/Span</i> to set the center value and the span of the sweep.
Start/Stop Start, Stop, Step-size, Pts/decade, Num. of pts.	Start Stop Step Dec Lin	Select the <i>Start/Stop</i> option to sweep based on start, stop, step-size and number of points. <i>Step-size</i> is <i>Pts./decade</i> for a Log sweep. - Start—the start point of a sweep - Stop—the stop point of a sweep - Step-size—the increments at which the sweep is conducted - <i>Pts./decade</i> —number of points per decade - Num. of pts.—the number of points over which sweep is conducted
Center/Span Center, Span, Step-size, Pts./decade, Num. of pts.	Center Span Step Dec Lin	Select the <i>Center/Span</i> option to sweep based on center and span, points per decade and number of points. <i>Pts./decade</i> is <i>Step-size</i> for a Linear sweep. - Center—the center point of a sweep - Span—the span of a sweep - Step-size—the increments at which the sweep is conducted - <i>Pts./decade</i> —number of points per decade - Num. of pts.—the number of points over which sweep is conducted
Note: Changes to any of the Start, Stop, etc. fields causes the remaining fields to be recalculated automatically.		
Use sweep plan	SweepPlan	Enables use of an existing sweep plan component ( <i>SweepPlan</i> ). Select this option and enter the name of the plan or select it from the drop-down list.

## Setting up a Sweep in RFDE

Setting up the sweep portion of the simulation consists of three basic parts:

- Enabling the Parameter Sweep.
- Specifying the parameter type.
- Specifying a sweep plan.

To shorten simulation time in any parameter sweep, select a start point where convergence is easy, and vary the parameter gradually. When selecting a start point, if you find that convergence is easier at one end of a sweep and harder at the other, use the easy end as the start of the sweep. Varying the parameter gradually yields better estimates for the next simulation, and achieves convergence more rapidly than if the parameter is changed abruptly. However, too small a step can lead to a prohibitively large number of sweep points, so be careful. For additional information about parameter sweep techniques, see [“About Preparing Sweeps” on page 1-20](#).

In the following table, names used in netlists appear under *Parameter Name*.

**Table 1-3. RFDE DC Simulation Parameter Sweep Parameters**

<b>Setup Dialog Name</b>	<b>Parameter Name</b>	<b>Description</b>
<b>Parameter Type</b>		
Design Variable		Click Select to choose a variable name from the Select Design Variable form. The list contains variables set up in the Editing Design Variables form. You can also enter a name in the Variable Name field. However, the variable must exist in the design for a successful simulation.
Temperature		<i>temp</i> is automatically selected because it is a reserved variable name.
Component Parameter		Click Select and choose a component from the schematic. In the Select Component Parameter form, select the parameter to be swept, then click OK. You can also enter the names in the Component Name and Parameter Name fields. However, the component and parameter must exist in the design for a successful simulation.
Model Parameter		Enter the name of a defined model and the name of the model parameter to be swept.
<b>Sweep Plan</b>		
<i>Choose one sweep range:</i>		
Start-Stop Start, Stop		Sets the Start and Stop values of the sweep - Start—the start point of the sweep - Stop—the stop point of the sweep
Center-Span Center, Span		Sets the Center value and a Span of the sweep. - Center—the center point of a sweep - Span—the span of a sweep
<i>Choose one sweep type:</i>		
Linear Step Size, Number of Steps		Enables sweeping a range of values based on a linear increment. Set the increment with Step Size <i>or</i> Number of Steps. Use Additional Points to add specific values. - Step Size—the increments at which the sweep is conducted - Number of Steps—the number of points over which sweep is conducted
Logarithmic Points Per Decade, Number of Steps		Enables sweeping a range of values based on a logarithmic increment. Set the increment with Points Per Decade <i>or</i> Number of Steps. Use Additional Points to add specific values. - Points Per Decade—the number of points per decade. - Number of Steps—the number of points over which sweep is conducted
Points Only		Enables simulation at specific values for the parameter. Enter values in the Specific Points field with a space between each one.
Additional Points		Additional Points - When sweep type is Linear or Logarithmic, click this option to enter specific values to include in the sweep range. Enter values with a space between each one.

## About Preparing Sweeps

In ADS and RFDE, the DC simulation setup dialog enables you to sweep a parameter. You can sweep a parameter using the internal sweep or use the ParamSweep component for an external sweep. The internal parameter sweep is recommended over an external parameter sweep when only one parameter is swept. To improve the convergence process during such an internally swept DC bias simulation, the simulator uses the results from the previous sweep point as an initial guess for the next sweep point and employs a sophisticated arc-length algorithm. In many cases, this makes for a faster and more robust simulation than one done with an external ParamSweep component. The latter simulation simply repeats the DC simulation at each sweep point, with no arc-length continuation. When multiple parameters are swept, the internal parameter sweep can be employed for one parameter and an external parameter sweep must be used for the remainder of the parameters.

The results of the swept DC bias simulation are saved in a binary form in a temporary file. This file is used to post-process selected measurements. Because this file contains *all* of the results, new measurements can be specified and displayed without the need for a subsequent simulation.

## Defining Simulation Parameters

Defining the simulation parameters consists of three basic parts:

- Specifying the desired level of detail in the simulation status summary.
- Specifying the amount of device operating-point information to save in ADS.
- Optionally, choosing to save all solutions to the dataset.

---

**Note** Advanced simulation parameters are accessible with this group. However, as a result of the improvements made to the DC simulation algorithm, it is extremely unlikely that the default values need to be modified. *You are strongly encouraged to leave the advanced parameters set to their default values.* If you encounter a circuit for which a DC analysis does not converge using the default values, or you find it necessary to change the value of any of these parameters, please contact Agilent EEs of Technical Support. See [“Defining Advanced Simulation Parameters” on page 1-23](#) for details about these parameters.

---

**Caution** In ADS and RFDE, simulator parameters saved in previous releases (in ADS design files, or in RFDE state files) are supported in later releases. The advanced simulation parameters saved prior to and opened in ADS/RFDE 2005A are recognized and populated in the simulation setup dialog box. However, due to the improvement in robustness and speed of the default DC simulation algorithm the user-defined values are disabled, and factory-defined default values are used. *Changing these default values is not recommended.* However, if you find it necessary to restore the original user-defined values, you must manually enable *Advanced Settings* to restore them.

---

In the following table, names used in netlists and ADS schematics appear under *Parameter Name*.

Table 1-4. DC Simulation Parameters

Setup Dialog Name	Parameter Name	Description
Levels (in ADS) Annotation (in RFDE) Enables you to set the level of detail in the simulation status report.		
Status level	StatusLevel	Prints simulation information in the Status/Summary part of the Message Window. - 0 reports little or no information, depending on the simulation engine. - 1 and 2 yield more detail. - Use 3 and 4 sparingly since they increase process size and simulation times considerably.  The type of information printed may include the sum of the current errors at each circuit node, whether convergence is achieved, resource usage, and where the dataset is saved. The amount and type of information depends on the status level value and the type of simulation.
Device operating point level (in ADS)	DevOpPtLevel	Options to save device operating-point information for most active devices and some linear devices in the circuit to the dataset. In ADS, if this simulation performs more than one DC analysis (from multiple DC controllers), the device operating point data for all DC analyses will be saved, not just the last one. Default setting is None. (For RFDE see <a href="#">“Saving Annotation Data in RFDE” on page 1-14.</a> )
None	None (0)	No information is saved.
Brief	Brief (2)	Saves device currents, power, and some linearized device parameters.
Detailed	Detailed (4)	Saves the DC operating point values which include the device's currents, power, voltages, and linearized device parameters.
Output solutions		
Output solutions at all steps	OutputAllSolns	Instructs the simulator to save all solutions in the dataset. When the simulator is required to use points between steps in order to converge, the resulting information is stored and can subsequently be used for more detailed analysis.
Advanced (in ADS)  Advanced Settings (in RFDE)		In ADS, click Advanced on Parameters tab, and enable Advanced Settings to set these parameters. In RFDE, click Advanced Settings to access these parameters.  For parameter descriptions, see <a href="#">“Defining Advanced Simulation Parameters” on page 1-23</a>

## Defining Advanced Simulation Parameters

The stand-alone DC simulator's sole role is to do a DC analysis. All other simulators such as AC, S-parameter, transient, harmonic balance, and circuit envelope do an initial DC analysis as their first step. The advanced simulation parameters are used for controlling a stand-alone DC simulation. For information about setting DC convergence parameters for these analyses, see *Simulation Basics > Using the Simulator Options > Setting Convergence Options* in the *Using Circuit Simulators* documentation.

The robustness and speed of the default DC analysis algorithm has been significantly improved in ADS/RFDE 2005A. All DC analyses with factory-default settings are expected to converge to the correct solution with near-optimal speed. This means that it is extremely unlikely that any of the following advanced simulation parameters must be altered:

*ConvMode*

*ArcMaxStep*

*MaxDeltaV*

*ArcLevelMaxStep*

*MaxIters*

*ArcMinValue*

*MaxStepRatio*

*ArcMaxValue*

*MaxShrinkage*

*LimitingMode*

In the following table, names used in netlists and ADS schematics appear under *Parameter Name*.

**Table 1-5. DC Simulation Advanced Settings Parameters**

Setup Dialog Name	Parameter Name	Description
Advanced Settings		Click Advanced Settings to set these parameters.
Max. Delta V	MaxDeltaV	Maximum change in node voltage per iteration. If no value is specified, the default value is four times the thermal voltage, or approximately 0.1 V.
Max. Iterations	MaxItrs	Maximum number of iterations to be performed. The simulation will iterate until it converges, an error occurs, or this limit is reached.
Mode	ConvMode	Offers a choice between different convergence algorithms. Note that the convergence algorithm selected here applies to this DC simulation only. You can choose from these same convergence algorithms using the simulator options to apply them to all analyses performed for this design.
Auto sequence	0	Default convergence mode. Cycles through various algorithms and parameter values and has been optimized for both robustness and speed. Should converge for all circuits, and is therefore strongly recommended over all other convergence modes.
Newton-Raphson	3	Iterative process that terminates when the sum of the currents into each node equals zero at each node, and the node voltages converge. Used by other convergence modes.
Forward source-level sweep	4	Sets all DC sources to zero and then gradually sweeps them to their full values. The source steps are determined via homotopy/continuation methods.
Rshunt sweep	5	Inserts a small resistor from each node to ground and then sweeps this value to infinity.
Reverse source-level sweep	6	Rarely used, but available for those few cases where it is necessary. Similar to Forward source-level sweep, except in the reverse direction. Use Reverse source-level sweep when Forward source-level sweep returns an "out of bounds" error. This error indicates that there is a negative resistance in the circuit when all the DC sources are zero. This is a rare situation but can occur with ideal models of oscillators, such as those described by the van der Pol equation.
Hybrid solver	7	Combination of various algorithms. Starts with Forward source-level sweep with the source steps determined via heuristics. If this fails, Forward source-level sweep with the source steps determined via homotopy/continuation methods is attempted. If this fails, Reverse source-level sweep with the source steps determined via homotopy/continuation methods is attempted. If this fails, Rshunt sweep is attempted. If this fails, Gmin relaxation, where a 1 Mohm resistor is inserted from each node to ground and then swept to infinity, is attempted.
Pseudo transient	8	Variant of the source stepping algorithm. Performs a transient simulation on a pseudo circuit derived from the original circuit. The transition from the zero solution to the final solution is of no interest in this analysis, so the truncation error is ignored and the timestep is taken as large as possible. After this pseudo transient analysis, a Newton-Raphson analysis is performed with the pseudo transient solution as the initial guess. If this fails, a Newton-Raphson analysis with Gmins of 1e-12 Siemens inserted from each node to ground is attempted. If this succeeds, the Gmins are removed and a Newton-Raphson analysis with the Gmin solution as the initial guess is attempted.

Table 1-5. DC Simulation Advanced Settings Parameters

Setup Dialog Name	Parameter Name	Description
Arc Max Step	ArcMaxStep	Limits the maximum size of the arc-length step during arc-length continuation. During arc-length continuation, the arc-length is increased in steps. The step size is calculated automatically for each problem. However, if ArcMaxStep is specified and is nonzero, it will define an upper limit for the size of the arc-length step. The default is 0, meaning there is no upper limit for the arc-length step.
Arc Level Max Step	ArcLevelMaxStep	Limits the maximum arc-length step size for source-level continuation. The default is 0 which means there is no limit for the arc-length step.
Arc Min Value	ArcMinValue	Set relative to ArcMaxValue. ArcMinValue determines the lower limit that is allowed for the continuation parameter $p$ during the simulation. During arc-length continuation, $p$ can trace a complicated manifold and its value can vary non-monotonically. ArcMinValue specifies a lower bound for $p$ such that if during the arc-length continuation $p$ becomes smaller than ArcMinValue, the simulation is considered to have failed to converge. The default is $p_{min} \cdot \text{delta}$ , where $\text{delta}$ is $p_{max} - p_{min}$ where $p_{min}$ is the lower end of the parameter sweep and $p_{max}$ is the upper end of the parameter sweep.
Arc Max Value	ArcMaxValue	Set relative to ArcMinValue. ArcMaxValue determines the upper limit that is allowed for the continuation parameter $p$ during the simulation. During arc-length continuation, $p$ can trace a complicated manifold and its value can vary non-monotonically. ArcMaxValue specifies an upper bound for $p$ such that if during the arc-length continuation $p$ becomes greater than ArcMaxValue, the simulation is considered to have failed to converge. The default is $p_{max} \cdot \text{delta}$ , where $\text{delta}$ is $p_{max} - p_{min}$ where $p_{min}$ is the lower end of the parameter sweep and $p_{max}$ is the upper end of the parameter sweep.
Max Step Ratio	MaxStepRatio	Controls the maximum number of continuation steps. The default is 100.
Max Shrinkage	MaxShrinkage	Controls the minimum size of the arc-length step. The default is 1e-5.
Limiting Mode	LimitingMode	Sets the type of limiting done on the changes of nodes at each iteration.
Global Element Compression	0	Limits the changes at the nonlinear nodes with a log function at each iteration when the changes exceed the internally determined value.
Global Device-based Limiting	1	Performs limiting on the changes at each iteration for the nonlinear components.
Dynamic Element Compression	2	Limits the changes at the nonlinear nodes with a log function at each iteration when the changes exceed the internally determined value.
Dynamic Vector Compression	3	Limits the changes of all nodes with a log function at each iteration when the changes exceed the internally determined value.
Global Vector Compression	4	Limits the changes of all nodes with a log function at each iteration when the changes exceed the internally determined value.
Global Vector Scaling	5	Scales the changes of the node at each iteration with internally determined scale factor.
No Limiting	6	No limiting will be done on the changes of all the nodes at each iteration.

## Additional Parameters

The following table includes additional parameters that you may find useful.

Table 1-6. Additional Parameters

Setup Dialog Name	Parameter Name	Description
Optimization		
Compute Sensitivities Using Finite Difference	UseFiniteDiff	<p>Perform sensitivity analysis optimization using the Finite Difference approximation method. This requires N+1 circuit simulations, where N is the number of optimization variables.</p> <p>To set this parameter in ADS, select the parameter name on the Display tab, then enter the value directly on the schematic. Allowed values: <i>yes</i> and <i>no</i>.</p>
Other		<p>Check this box to enable access to hidden parameters. The text field is enabled to allow assigning values to the parameters. The format is <code>Other=HiddenParameter1=value1 HiddenParameter2=value2. . .</code></p> <p>Hidden parameters typically are used when troubleshooting convergence problems.</p> <p>To set this parameter in ADS, select the parameter name on the Display tab, then enter the value directly on the schematic.</p>

# Theory of Operation

This section discusses the theory of operation for DC simulation. It can help you understand the underlying technology. Typical users should not need to examine this material.

## Simulation Basics

The simulators compute the response of a given circuit to a particular stimulus by converting, based on certain assumptions, a system of nonlinear ordinary differential circuit equations into a system of nonlinear algebraic equations and then solving them numerically. The various simulators convert ordinary differential equations to algebraic equations differently and use different numerical techniques for solving the resulting algebraic equations, leading to the many different simulator flavors (DC, AC, S-parameter, transient, harmonic balance, circuit envelope). For example, the DC and the harmonic balance simulators treat the  $d/dt$  operator differently, leading to different algebraic equations. The numerical simulation techniques rely on various iterative processes to achieve mathematical convergence toward an equilibrium point in the nonlinear algebraic equations that describe the circuit. Once this equilibrium point is reached to within certain tolerances, a solution is said to have been found.

The specific assumptions for the DC simulator are described in [“Simulation Assumptions” on page 1-27](#).

## Simulation Assumptions

DC voltages and currents are signals of zero frequency. The simulator uses this concept when performing a DC simulation, and the following conditions apply:

- Independent sources are constant valued.
- Linear elements are replaced by their (real) conductances at zero frequency.
- Capacitors, microstrip gaps, AC coupled lines, and similar items are treated as open circuits.
- Inductors, conductive discontinuities, and similar items are treated as short circuits.
- Time-derivatives are constant (zero).
- Transmission lines are replaced by DC conductance values calculated from their length, cross-sectional area, and conductivity.

- Scattering parameter (S-parameter) files must include zero frequency data to operate properly at DC (this is also required for harmonic balance analysis). Otherwise, the simulator extrapolates each S-parameter for the zero-frequency case, and uses the real part as the DC response.
- The simulator has built-in safeguards against nodes that are DC-isolated (that have no DC path to ground), as well as against DC source-inductor loops. Nevertheless, try to avoid these conditions.

## For More Details

The descriptions of the various convergence modes in [Table 1-5](#) mention techniques such as the Newton-Raphson algorithm, source-level sweeping, arc-length continuation, Gmin relaxation, and pseudo-transient analysis. For more details about these and related techniques, see the publications listed in [“References” on page 1-36](#).

# Troubleshooting a Simulation

This section presents suggestions for using this simulator and improving the accuracy of results.

The robustness and speed of the default DC analysis algorithm has been significantly improved in ADS/RFDE 2005A. All DC analyses with factory-default settings are expected to converge to the correct solution with near-optimal speed. This means that it is extremely unlikely that any of the following advanced simulation parameters must be altered:

<i>ConvMode</i>	<i>ArcMaxStep</i>
<i>MaxDeltaV</i>	<i>ArcLevelMaxStep</i>
<i>MaxIters</i>	<i>ArcMinValue</i>
<i>MaxStepRatio</i>	<i>ArcMaxValue</i>
<i>MaxShrinkage</i>	<i>LimitingMode</i>

---

**Note** As a result of the improvements made to the DC simulation algorithm, it is extremely unlikely that the factory-default values for the advanced simulation parameters need to be modified. *You are strongly encouraged to leave these parameters set to their default values.* If you encounter a circuit for which a DC analysis does not converge using the default values, or you find it necessary to change the value of any of these parameters, please contact Agilent EEs of Technical Support. See [“Defining Advanced Simulation Parameters” on page 1-23](#) for details about these parameters.

---

Use the following checklist to locate and fix problems. For details about specific problems, see the sections that follow.

- Turn on *Issue warnings (GiveAllWarnings)* and increase *Maximum number of warnings (MaxWarnings)* in the *Options* component.
- Read all error and warning messages from the Simulation/Synthesis Messages window and make corrections accordingly.
- Read all status information from the Status/Summary window (select *Show Complete Status Message* under Window tab) and make corrections accordingly.

- Check your setup for common errors (see [“Common Setup Errors” on page 1-30](#)).
- Loosen the simulation Options parameters for absolute and/or relative simulation tolerances (see [“Adjusting Tolerance Parameters” on page 1-31](#)).
- Turn on/off the Topology Checker (on by default) using the simulation Options parameters (see [“Topology Check and Correction” on page 1-32](#)).
- Check if you have created a circuit with no solution (see [“Impossible Circuits” on page 1-36](#)).
- Check if I<sub>max</sub> should be increased (see [“Model Parameter I<sub>max</sub> is Too Small” on page 1-36](#)).

## Common Setup Errors

If your circuit does not converge, or if it converges but your DC I-V curves appear different than expected, it can be due to one or more of the following reasons:

- Check component and model parameter values and units, especially small series resistor values in model parameters.
- Check if your independent swept parameters are swapped on the grid display. For example, if SRC1 sweeps a transistor gate voltage and SRC2 sweeps a drain voltage, and if the current is displayed with SRC1 (gate voltage) on the  $x$ -axis, these traces will not look like typical DC I-V traces. Exchange the order of the swept variables SRC1 and SRC2 on the grid.
- Check if you are sweeping one source with large variations and another with small variations. For example, say that you are sweeping the base current and the collector voltage of a transistor by connecting a DC current source at the base and a DC voltage source at the collector. Furthermore, say that you are using SRC1 to sweep the base current and SRC2 to sweep the collector voltage. A DC bias-current sweep typically involves very small increments—on the order of microamperes, for example. Inside the program, the SRC1 sweep always occurs inside the SRC2 sweep, so the net effect is that the base current is changing more frequently. Because of the small increments in the current sweep, proceeding from one current value to the next may not change the circuit state (operating point) in the simulator. This causes the program to converge to the same solution as the previous current value, resulting in jagged I-V curves.

- Check if you are measuring the DC current in the wrong direction. By default, positive current through a two-terminal element, such as a resistor or ammeter, is measured as flowing from pin 1 to pin 2. To measure current in the opposite direction, you can edit the schematic by changing the element's orientation, or specify the current at a different pin, or change the orientation of the current probe.
- If you are having convergence problems during a swept DC simulation, reduce the step size of the parameter being swept.

## Adjusting Tolerance Parameters

Several tolerance parameters can be changed so that simulations are more likely to converge, although at the cost of a somewhat less accurate result. These parameters include:

- Current relative tolerance ( $I\_RelToI$ )
- Voltage relative tolerance ( $V\_RelToI$ )
- Current absolute tolerance ( $I\_AbsToI$ )
- Voltage absolute tolerance ( $V\_AbsToI$ )

The equations that are used to check for convergence apply to all analysis types, not just DC. In ADS, you can change these parameters in the *Options* component on the Convergence tab. In RFDE, you can change these parameters in the Simulator Options form.

## Topology Check and Correction

A topology check is performed before a simulation is run. This is the simulator's default setting. The topology check finds common circuit errors that would lead to more serious errors later in the simulation. These errors are reported in the Simulation/Synthesis Messages window. The topology check verifies the existence of the following conditions, some of which are corrected automatically by the simulator if the topology checker is turned on:

- There are no degenerate nodes.

At least two components must be connected to each node. Degenerate nodes have only one device attached to them, making it difficult for the simulator to solve the circuit. If the topology checker is on, the simulator automatically corrects this problem by inserting a large resistor from the offending node to ground.

- There is a DC path from every node to ground.

If the DC circuit simulation is to succeed, all circuit nodes must have a DC path to ground. This eliminates nodes that float at an undefined DC voltage value and allows the DC simulator to find the bias point of all nodes in the circuit. This error is commonly caused by two capacitors in series. If the topology checker is on, the simulator corrects this problem automatically by inserting a large resistor from the offending node to ground. This adds a DC path to ground without affecting the circuit's performance.

- There are no devices in series with current sources.

Devices in series with a current source create problems during DC simulation. This should be corrected manually.

- There are no loops of DC shorts, or voltage or current sources.

It is possible to create a circuit containing short-circuited DC loops that cannot be handled by the DC simulation. For instance, if two ideal inductors are connected in parallel, they create a DC short-circuit loop. The DC current in this loop is undefined; it can have any value without affecting the circuit solution. This should be corrected manually.

- There are no devices parallel to voltage sources.

Devices parallel to a voltage source create problems during DC simulation. This should be corrected manually.

The following figures show examples of the topology conditions checked when the topology checker is turned on, the parameter's default setting (*TopologyCheck=yes*). The simulator corrects the errors shown in [Figure 1-5](#) by inserting a large resistor from the offending node to ground. However, the simulator cannot correct the errors shown in [Figure 1-6](#), which you should correct manually. Additionally, the topology checker does not check and correct any of these conditions through transmission lines.

By default a summary of the topological problems found is printed to the Simulation/Synthesis Messages window if *TopologyCheck=yes*. To see a list of all the nodes that have topological problems, set the *TopologyCheckMessages* parameter to *verbose*. To set this mode in ADS, see the Misc tab of the Options component's setup dialog box, and choose *Verbose* in the Topology Checker section. In RFDE, in the Simulator Options dialog box, choose *Verbose* for Topology Check Messages.

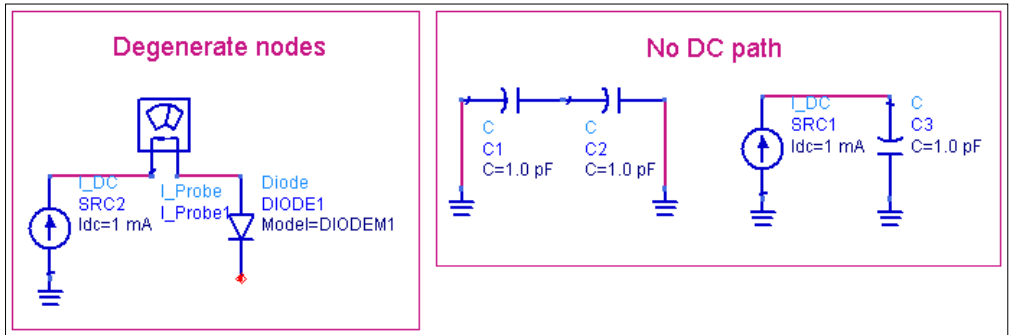


Figure 1-5. Common circuit errors corrected during topology check

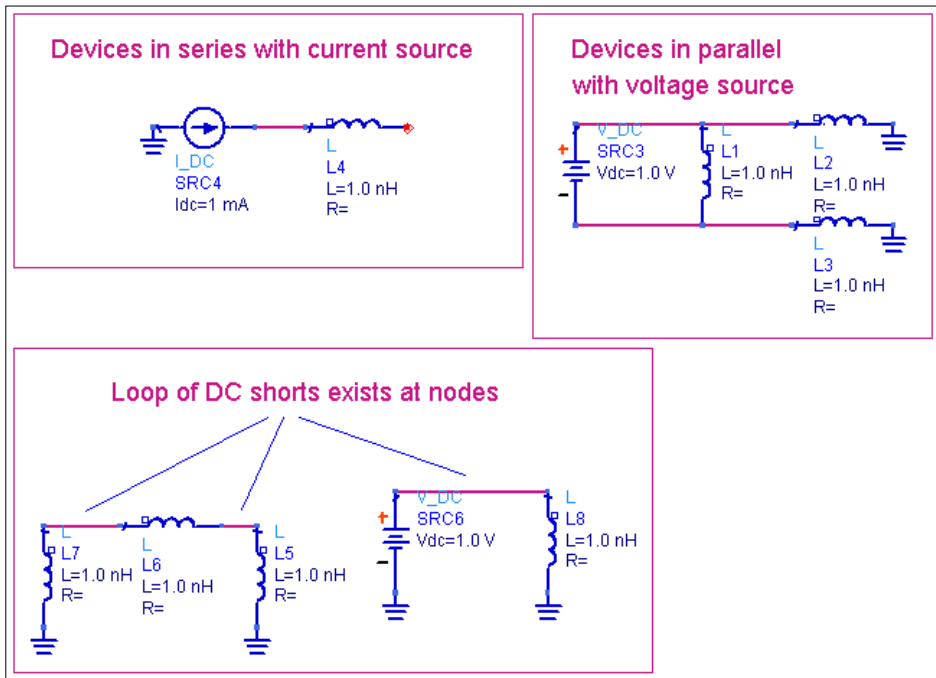


Figure 1-6. Common circuit errors *not* corrected during topology check

If the circuit fails to simulate due to topological problems, turn off the topology checker. In ADS, on the Options component's Misc tab, disable the *Perform topology check and correction* option. In RFDE, disable the *Perform topology check* in the Simulator Options dialog box. This will turn off all topology checking, and simulations will proceed directly to the DC simulation stage. The DC simulator will speed up simulations slightly, which may be helpful in rare cases where it is certain that the DC simulation will succeed despite the topology.

---

**Note** Errors found by the topology checker almost always lead to DC simulation failures, so turning off the topology checker is not recommended.

---

If you have existing circuits in which there are many topological problems, turning on the topology checker could significantly speed up the simulation. In this case the topology checker corrects the common topological problems before the circuit is simulated.

---

## Checking Nodes and Pins

If the topology check fails, the offending nodes are highlighted in the Simulation/Synthesis Messages window. Choose *Simulate > Highlight Node* to identify these nodes.

## Unconnected Wire Errors

In ADS, in the case of a circuit with a wire that is missing, a message such as the following can appear in the Simulation/Synthesis Messages window during a simulation:

*Warning detected by HPEESOFSSIM during DC analysis 'DC1'.  
Circuit as given has no unique solution.  
A virtual resistance of 1 TOhms was added to each node.*

In the case of unconnected wires or pins, choose *Tools > Check Representation*, then select *Open connections* and click *OK*. All unconnected wires and components are highlighted, making them easily visible on the screen. Connect these to the correct circuit node.

## Port/Pin Mismatches

In ADS, to check for mismatches between ports and named pins on an underlying schematic, use the Check Representation feature. Choose *Tools > Check Representation*, then select *Port/Pin mismatches* and click *OK*. In a circuit without a topology problem, the simulator reports the following:

*Unconnected pins: 0  
Port/Pin mismatches: 0*

Where a wire is missing between two pins, for example, the components and the disconnected pins will be highlighted in red on the schematic, and a message such as the following will appear, indicating the coordinates of the connectors:

*Unconnected pins: 2  
I\_DC SRC1, pin 2 (2.500,-0.125)  
BJT\_NPN BJT1, pin 2 (3.250,0.375)*

To clear the highlighting, choose *View > Clear Highlighting*.

---

**Note** Always ensure that there is no duplication of port numbers.

---

## Impossible Circuits

A circuit can fail to have a solution. Examples of such impossible circuits are circuits with two DC current sources (or, perhaps, current-mirror circuits) in series. The simulator attempts to analyze these circuits by adding a large resistor in parallel with every node in the circuit. If the DC simulation then succeeds, the resistor value is increased as high as possible (up to 1 Tohm). This allows the DC simulation to proceed, and usually to succeed. However, the offending circuit node or nodes often cause problems with a subsequent harmonic balance simulation because the harmonic balance simulator does not add these resistors. It is best to find the problem node and manually add a DC path (such as a large resistor) to ground.

If virtual resistors were added to each node during DC simulation but the simulation still failed, look for circuit problems such as loops of inductors and voltage sources.

## Model Parameter $I_{max}$ is Too Small

If the  $RHS\_NORM$  oscillates between/among iterations, the reason could be that the diode's p-n junction in this circuit conducts more than 1 A. The default values of  $I_{max}$  for nonlinear semiconductor devices are around 1 A. It is possible that all device's p-n junction currents in a circuit do not exceed  $I_{max}$  after the circuit converges but have problems converging during the iteration process because of the small  $I_{max}$ . The remedy is to increase nonlinear device model parameter  $I_{max}$  to 1e3. For BSIM3, the model parameter is  $I_{jth}$ .

## References

- [1] K.S. Kundert, "The Designer's Guide to SPICE and Spectre," Kluwer Academic Publishers, Boston, MA, USA, 1995.
- [2] K.S. Kundert, J.K. White, and A. Sangiovanni-Vincentelli, "Steady-State Method for Simulating Analog and Microwave Circuits," Kluwer Academic Publishers, Boston, MA, USA, 1990.
- [3] L.W. Nagel, "SPICE2: A Computer Program to Simulate Semiconductor Circuits," Ph.D. Dissertation, Electronics Research Laboratory, College of Engineering, University of California, Berkeley, CA, USA, 1975.

# Index

## A

- absolute tolerances, 1-31
- advanced parameters
  - described, 1-23
- analysis technique, 1-4
- annotation data (in RFDE)
  - adding data to schematic, 1-14
  - described, 1-14
  - saving, 1-14
  - setting level, 1-22
- ArcLevelMaxStep, 1-25
- ArcMaxStep, 1-25
- ArcMaxValue, 1-25
- ArcMinValue, 1-25

## B

- binning
  - defined, 1-8

## C

- convergence problems
  - impossible circuits, 1-36
  - tolerance parameters, 1-31
  - troubleshooting, 1-29
- ConvMode, 1-24
- current absolute tolerance, 1-31
- current relative tolerance, 1-31

## D

- DC I-V curves, 1-30
- DC simulation
  - and other simulations, 1-3
  - degenerate nodes errors, 1-32
  - described, 1-1
  - impossible circuits, 1-36
  - open-circuit errors, 1-32
  - parameters, 1-14-1-26
  - performing, 1-3
  - problems with I-V curves, 1-30
  - purpose, 1-3
  - selecting in ADS, 1-1
  - selecting in RFDE, 1-1
  - short-circuit errors, 1-32
  - Simulator Options tolerance parameters, 1-31

- single-point simulation, 1-5
- speed and robustness, 1-23, 1-29
- tolerance parameters, 1-31
- topology error checking, 1-32
- troubleshooting, 1-29
- degenerate nodes topology errors, 1-32
- description of DC simulation, 1-1
- DevOpPtLevel, 1-15, 1-22

## G

- GiveAllWarnings, 1-29

## I

- I-V curves, 1-30

## L

- levels (in ADS), 1-22
- license requirement, 1-3
- LimitingMode, 1-25

## M

- MaxDeltaV, 1-24
- MaxIters, 1-24
- MaxShrinkage, 1-25
- MaxStepRatio, 1-25
- MaxWarnings, 1-29

## O

- open-circuit topology errors, 1-32
- Other hidden parameters, 1-26
- OutputAllSolns, 1-22

## P

- parameters, 1-14-1-26
  - advanced parameters, 1-23
  - annotation data (in RFDE), 1-14
  - ArcLevelMaxStep, 1-25
  - ArcMaxStep, 1-25
  - ArcMaxValue, 1-25
  - ArcMinValue, 1-25
  - ConvMode, 1-24
  - DevOpPtLevel, 1-15, 1-22
  - hidden, 1-26
  - LimitingMode, 1-25
  - MaxDeltaV, 1-24
  - MaxIters, 1-24

- MaxShrinkage, 1-25
- MaxStepRatio, 1-25
- Other, 1-26
- OutputAllSolns, 1-22
- SaveDCCurrentSolution, 1-15
- SaveDCVoltage, 1-15
- status report levels, 1-22
- StatusLevel, 1-22
- SweepPlan, 1-17
  - sweeps (in ADS), 1-16
  - sweeps (in RFDE), 1-18
- SweepVar, 1-17
- UseFiniteDiff, 1-26
- ParamSweep (in ADS), 1-20
- ports
  - pin mismatches, 1-35
- purpose of DC simulation, 1-3

## R

- relative tolerances, 1-31

## S

- SaveDCCurrentSolution, 1-15
- SaveDCVoltageSolution, 1-15
- setting up a simulation in ADS, 1-4
- setting up a simulation in RFDE, 1-4
- short-circuits
  - correcting topology errors, 1-32
- simulation parameters
  - DC simulation, 1-14-1-26
- simulations, other, 1-3
- Simulator Options
  - setting tolerance parameters, 1-31
- single-point simulation, 1-5
- StatusLevel, 1-22
- SweepPlan, 1-17
  - sweeps (in ADS), 1-16
  - sweeps (in RFDE), 1-18
- SweepVar, 1-17

## T

- tolerance parameters, 1-31
  - and DC simulation, 1-31
  - in Simulator Options, 1-31
- topology checks
  - corrections, 1-32
  - degenerate nodes, 1-32

- devices in parallel with voltage sources,
  - 1-32
- devices in series with current sources,
  - 1-32
- examples, 1-33
- open circuits, 1-32
- short circuits, 1-32
- troubleshooting
  - DC simulation, 1-29

## U

- UseFiniteDiff, 1-26

## V

- voltage absolute tolerance, 1-31
- voltage relative tolerance, 1-31