Agilent 85190A
IC-CAP 2008

Nonlinear Device Models
Volume 2

Agilent Technologies

Notices

Edition

Acknowledgments

UNIX ® is a registered trademark of the Open Group.

Windows ®, MS Windows ® and Windows NT ® are U.S. registered trademarks of Microsoft Corporation.

Errata

The IC-CAP 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

Technology Licenses

Restricted Rights Legend

Safety Notices

CAUTION

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

WARNING

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

Contents

1

Agilent Root FET Model Generator

This chapter documents the Agilent 85191A Agilent Root FET model generator software. It provides an example procedure that takes you step-by-step through a complete data acquisition and model generation.

The complete procedure includes all the instructions needed to generate a new device model by modifying the variable values in factory default tables. After you have set these variables, subsequent procedures are much faster.

Agilent Technologies

Model Description

The Agilent Root FET model is a data-based model, rather than a physical or empirical model. It generates a device-specific large-signal model directly from measured DC and small-signal S-parameter data, that accurately represents the terminal characteristics of the device. The model is constructed without circuit simulation or optimization. Use of S-parameter as well as DC data results in a model that can accurately predict nonlinear device performance with frequency dispersion, as a model simulated with a single set of nonlinear current functions cannot. The Agilent Root FET model predicts the device performance as a function of bias, power level, and frequency. It allows circuits containing devices for which measured data exists—but for which there are no good physical or empirical models—to be simulated.

The Agilent Root FET model works for MESFETs and HEMTs (MODFETs). It is scalable to allow simulation of devices with different geometries based on the characterization of a single device. The present implementation is used for positive drain-source bias (Vds ≥ 0), which corresponds to the typical operating region.

Based on gate-current and power-dissipation device compliances you provide in software, the data acquisition system calculates the safe operating range for the device. Within this region, the model takes data adaptively at multiple bias points, depending on the specific nonlinearities of the device under test. Measurements are densely spaced in the most nonlinear regions, such as at the knees of the IV curves and the onset of breakdown, and less densely spaced in the linear saturation region.

The model data set of internodal nonlinear current and charge components at each bias point is then mathematically generated and stored. The model is thus generated without the usual need for simulation and optimization as required by all previous empirical models. The Root FET model may thus capture nonlinear device behavior for any MESFET device for which measured data can be taken but for which there are no good physical or empirical models.

The data acquisition system sets the DC bias levels and controls the system hardware to take the DC and S-parameter measurements, based on your inputs. The model generator processes the measured data and generates the lookup tables required by a circuit simulator. It produces data files that are directly readable by the Agilent RF and Microwave Design System (MDS). Both IC-CAP and MDS can be run simultaneously in the X Windows environment. During a simulation, the tabular state-function data from the generated Agilent Root FET model is interpolated using multidimensional spline functions to emulate the terminal characteristics of the device.

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP.

Getting Started in IC-CAP

This example procedure assumes the following:

- The system hardware has been set up and switched on according to the instructions under

    - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

    - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

    - Wiltron 360-based system—Refer to your Wiltron documentation

- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.

- The IC-CAP software has been properly installed on the computer.

- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:

    - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

    - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename the SMUs to correspond with the FET terminals.

Rename a medium-power SMU such as MPSMU1 as *VG* (gate supply)

Rename a high-power SMU such as HPSMU3 as *VD* (drain supply)

Rename the *GNDU* unit (which must be connected to the device source) as *GND*.

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on opening the IC-CAP software and accessing the model file you need. Find the Agilent Root FET model file under:

   /examples/model_files/mesfet/HPRootFet.mdl

Then return to this chapter and continue with the procedure.

<table>
<tr><td>NOTE</td><td>Do not install the device yet. The procedure will instruct you to install it after extracting the measurement port resistances.</td></tr>
</table>

Before you continue with the IC-CAP procedures, make a new directory in an appropriate location to store the data and model files that will be generated. Give the directory a name that will associate it with the device you are going to model. Use the *cd* command to change to the new directory.

MNS Link

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP. The link is set up for MDS version B.07.00, and will need to be modified when MDS is updated. For the link to work, the path and environment variables need to be set correctly: *this is done at the factory for the path and environment of a typical MDS B.07.00 system. Only minor changes to reflect unusual MDS paths or updated MDS versions are likely to be required.* If you do need to modify your UNIX files to reset the MNS link, follow these steps to set the root MDS directory on your system and add $MDSROOT/mdsbin to the path:

For shell or kshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory to automatically set the environment variables:

   MDSROOT=/mds_b.07.00 (depending on the version and location of MDS)
   export MDSROOT
   PATH=$MDSROOT/mdsbin: "$PATH"
   export PATH

2 Also add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the MNS simulator:

   mns mns /mds_b.07.00/hp 85150/lib/mns800 "" CANNOT_PIPE

For cshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory:

   #Setup for mns simulator:

   setenv MDSROOT /mds_b.07.00
   setenv PATH $MDSROOT/mdsbin: "$PATH"

2 Add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the mns simulator:

   mns mns /mds_b.07.00/hp 85150/lib/mns800 "" CANNOT_PIPE

Using the Agilent Root FET Model

This chapter provides an example procedure that takes you step-by-step through a measurement and model generation. The device used in this example was a GaAs MESFET, and the device data sheet was used for information on device characteristics and compliance values. Values you use in generating your own device model will depend on the particular characteristics of your device: use its data sheet (if available) as a reference.

This example is provided as a starting point. It is a straightforward procedure that uses only a subset of IC-CAP's total capabilities. The complete procedure documented gives all the steps necessary to configure IC-CAP with the values required for preverification measurements, parasitic extractions, and the main data acquisition. Once a model has been developed for a particular device type, the built-in scaling feature allows you to apply it to any device with the same geometry and process. The modeling procedure then becomes more simple, and can be accomplished in a few easy steps.

NOTE   Although you will modify the variable values in factory default files during this procedure, do not under any circumstances change the *names* of any variables or setups. These names are used by the software in the data acquisition and model generation process.

The example procedure measures the FET under test in a common-source configuration.

The example procedure gives detailed instructions on how to:

- Initialize device parameters.

- Measure and extract the system measurement port series resistances.

- Make a DC measurement to preverify device DC performance (optional).

- Calibrate the network analyzer for both swept and CW measurements.

- Make an S-parameter measurement to preverify device performance at high frequencies (optional).

- Measure and extract the ideality factor.

- Measure and extract the device parasitic resistances and inductances.

- Set up and initiate the main data acquisition. Plot the measured data.

- Set up and initiate the model generation, first selecting a starting point for contour integration. Plot the calculated state functions.

- Verify the model against measured data.

- Store the data and model files for circuit simulation in Agilent MDS.

Where necessary, other chapters are cross-referenced at appropriate points in the procedure, so that you will have the information you need for system hardware setup and calibration at the proper times.

NOTE   Measurement values and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct results.

The procedure provides guidelines on setting values to measure your device. However, the values you use will be based on your knowledge of your device and on information in its data sheet, as well as on the guidelines presented here: they may or may not correspond with the example values.

The illustrations of plotted data are provided as visual examples of possible results. They are not intended to represent a single device, nor do they necessarily correspond with the values given in the procedure. The plots can be used to check the reasonableness of your own data, which should appear similar in shape but not necessarily coincident in values.

The DUTs and Setups

The Model window is the central access point for the measurement and model generation procedures. The Model window includes the DUT/Setup panel and the file tabs to access the Model Parameters table, the Model Variables table, and other IC-CAP features.

The illustration shows the DUT/Setup panel for this model. The DUTs (for example *pre_verify*), indicated by the inverted triangles, are groupings of similar measurement setups used to extract related model parameters. The setups (for example *idvd_vg*) contain the information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.
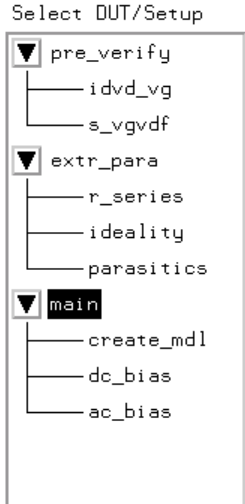
Figure 1   The DUT/Setup Panel

The MACRO list provides access to functions that combine certain groups of IC-CAP operations into single operations.

Initializing Device Parameters

You begin the procedure by entering certain key values for the device and the model generation, and giving the device an identifying name. For this you use the *init_parameters* macro. When this macro is executed, it also initializes internal program functions that are essential in the model generation process, and it provides annotations for the data plots.

1 Click the **Macros** tab, and a list of macros is displayed.

2 In the Select Macro list, select **init_parameters**, and **Execute**.

3 A series of dialog boxes is displayed, asking you to input appropriate values for your device under test. Each time you enter a value, click **OK** or press Enter.

4 Enter the extraction frequency. This is the CW frequency you will use for the data acquisition, and must be the same as the frequency of your CW calibration. If you know the fundamental frequency of anticipated operation, use that. If not, a good rule of thumb is to use a value such that $f_T/5 \leq f_{extract} \leq f_T/2$.

   f_T of a device is given by the equation: $f_T \approx \dfrac{gm}{2\pi C_G}$

5 Enter the forward conducting gate current in amps. This is typically 200 mA per mm of gate width.

6 Enter the reverse gate breakdown current in amps. This is typically 20 mA per mm of gate width.

7 Enter the number of gate fingers.

8 Enter the width of the device gate, in meters.

9 Enter a name to identify the device under test.

10 When you finish responding to the questions in the dialog boxes, the macro will close.

CAUTION   Do not edit the *init_parameters* macro. Especially, do not make any changes below the line (as instructed in the macro). The information below the line is *executable code*: modifying it can cause problems in the program.

Parameters Table

The illustration shows the Model Parameters table, with initial values for the extrinsic parasitic capacitances and inductances. The table is presented so that you can enter values for these extrinsic parasitics based on your knowledge of the device, or on estimates, or on measurements of the open fixture. In addition, you can set limits by entering minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values.

Param Name	Value	Min	Max
cpds.c	0.000		
cpgd.c	0.000		
cpgs.c	0.000		
ls_ext.l	0.000		
ld_ext.l	0.000		
lg_ext.l	0.000		

Figure 2   Model Parameters Table

To change a value in the Parameters table, move the mouse pointer over the current entry and click left to highlight it, then type in a new value.

The default for all the *c* values is 0.000. A reasonable capacitance value to enter for a pad on a wafer might be 50 fF.

The *l_ext.1* values are the bond inductances for the three terminals. The default value for all three is 0.000. If you know or can estimate the bond inductances for your device based on wire length or grid size, enter appropriate values.

Extracting the Measurement Port Series Resistances

This procedure need be performed *only* following initial installation of the system and if changes are made in the system hardware.

The procedure measures a short circuit at the normal device measurement interface plane. IC-CAP controls the DC source/monitor to force a gate current and monitor the gate voltage. The primary measurement is at port 1, and the port 2 measurement is synchronized to the port 1 measurement. You set the measurement parameters by defining the instrument states and the DC input values, as explained in the next pages.

The software calculates the series resistances from the outputs of the bias networks to the inputs of the device under test. The resistance values are then incorporated into the model file, and their effects are removed from the device response during the modeling process. (The default port resistance for both port 1 and port 2 is 0Ω.)

Note that the first part of the procedure differs for probe station and in-fixture measurements.

Setting Up for the Measurement

1 *If you are using a probe station*, position an impedance standard substrate (ISS) on the chuck. Place both the port 1 and port 2 probes on a short circuit standard.

2 *If you are using a fixture*, insert the short from the in-fixture calibration kit.

3 From the DUT/Setup panel select the r_series setup.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Set Integ Time to M (medium). Then return to this chapter and continue.

Setting the r_series Inputs

Use the following guidelines for setting the *ig* and id inputs (in most cases, the default settings should suffice):

Table 1   For the ig input

Set	To
Mode	I
To Node	*G* (gate)
From Node	GROUND
range	*Start* and *Stop* current values that define a current range within the limits of the bias network (see *Caution* below)
	# of Points sufficient to make several measurements across the current range
Compliance	As high as 5 to 10V. With a current input, compliance refers to voltage. You can set the compliance as high as 5 to 10V because the device is not yet connected.

The *Step Size* will be set automatically.

Table 2   For the id input

Set	To
To Node	D
Unit	VD
Compliance	same as ig
SweepType	*SYNC*
Master Sweep	*ig* (so that the *id* sweep is synchronized to the *ig* sweep, with a 0.0 Offset)
Ratio	*1.000* (so that the id sweep is synchronized to the *ig* sweep)

> **CAUTION**   The bias networks can be destroyed if IC-CAP measurement current values are set too high! Be sure you know the maximum current ratings of your bias networks, and do not set current values beyond these limits. (The standard bias networks for the Agilent 85122A and 85123A parameter extraction test systems are rated at 0.5 amp.)

- The *Unit* name must match the name of the SMU connected to the device gate (probably *VG*). This is also the SMU unit name set in the Hardware window. Refer to the appropriate chapter as follows:

    - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

    - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

- The default settings for the *Outputs* should not be changed.

Measuring and Extracting

1 Select **Measure/Simulate > Measure**.

2 When the measurement is complete, the Measuring light on the DC source/monitor goes out and the IC-CAP wristwatch icon goes away.

3 Select the **Extract/Optimize** tab.

4 Select **compute_rseries > Execute**. The transform calculates the port series resistances from the measured data and inserts the values into the model variable table as r1s and r2s.

Installing the Device

> **CAUTION**   Ground yourself with an antistatic wrist strap to reduce the chance of electrostatic discharge. (Note that the Agilent 85122 or 85123 system work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.)

Install the device carefully on the probe chuck or in the fixture. Be sure to handle the device as little as possible, to avoid damaging it. Once the device is in place, avoid bumping the test station.

CAUTION   Remove any high-intensity light sources such as microscope light before taking a measurement. One simple method of blocking light is to place an opaque box on the fixture directly above the device.

Preverifying the Device DC Data

This procedure uses the idvd_vg setup to measure drain current with respect to drain voltage, at several values of gate voltage. The drain voltage is swept from 0V to the upper limit of its normal operating range. This is an opportunity to take a first look at the device performance and explore its characteristics. This also lets you select the most appropriate measurement range for the main data acquisition and other measurements.

In addition, this is a quick way to verify that the device DC characteristics are good, before starting a complete data acquisition and model generation. The data from this measurement can also be used for later comparison with data predicted by the generated model. It is not necessary to do the preverification measurement every time you perform a model generation, especially if you are using a data sheet.

NOTE   If you do not wish to preverify either the DC or S-parameter performance of the device, you can skip ahead to the section "Measuring and Extracting the Ideality Factor" on page 39.

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the r_series measurement. Use the same settings.

Setting the idvd_vg Inputs

This procedure defines the input signals to be applied to the device under test for the DC preverification measurement. The actual settings you use will be dependent on the compliance limits for both the device and the SMUs/bias networks, and on the range of measurements you wish to take. Since this measurement is intended only to preverify the device, the values can be quite conservative and need not stress the device to the boundaries of its performance.

If you are measuring a series of devices of the same type and have already set the inputs, go directly to "Measuring and Plotting" on page 28. Otherwise, perform the following steps:

Select the **idvd_vg** setup.

Table 3   For the vd input

Set	To
Mode	V (voltage)
Compliance	No greater than the maximum breakdown current value for the device.(You set compliance values to limit SMU output voltage or current and thus prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used. With a voltage input, compliance refers to current)
Sweep Type	LIN (to provide a linear voltage sweep from start to stop voltage values)
range	Start at 0.0
	Stop at or below the upper limit of the device's normal operating range
	# of Points to a value sufficient to clearly display the results

Note: The Step Size is calculated automatically in software from the start/stop values and number of points

Table 4   For the vg input

Set	To
Mode	V (voltage)
Compliance	No greater than the maximum allowable gate current value (You set compliance values to limit SMU output voltage or current and thus prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used. With a voltage input, compliance refers to current)
Sweep Type	LIN (to provide a linear voltage sweep from start to stop voltage values)
range	Such that Start, Stop, and # of Points will yield three to five measured values (points), in a range from approximately Vgs=Vpinchoff to Vgs=0V. Set Start at a value close to Vpinchoff, but not so close that the curve will be difficult to discern. Set Stop to 0.

- The Unit names must conform to the names set in the Hardware window. Refer to the appropriate chapter as follows:

  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

  - "Performing Hardware Setup in IC-CAP" on page 617 Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

They must also correspond with the actual SMU connections from the Agilent 4142. Since the source is grounded, only the drain and gate inputs need to be set.

CAUTION   Do not set the value of Vg high enough to damage the device.

- The default settings for the Outputs should not be changed.

Measuring and Plotting

This procedure triggers the DC source/monitor measurement and monitors the results.

NOTE	Before making the DC measurement, manually put the network analyzer in hold sweep mode by pressing STIMULUS > MENU > MORE > HOLD.

1   Select the **Plots** tab.

2   Select **Display All** (to the left of the setup window). Two plots are displayed.

3   Select the **Measure/Simulate** tab, then select **Measure**. When the measurement is complete, the Measuring light on the DC source/monitor goes out, the IC-CAP wristwatch icon goes away, and the measured data is displayed in the plot windows.
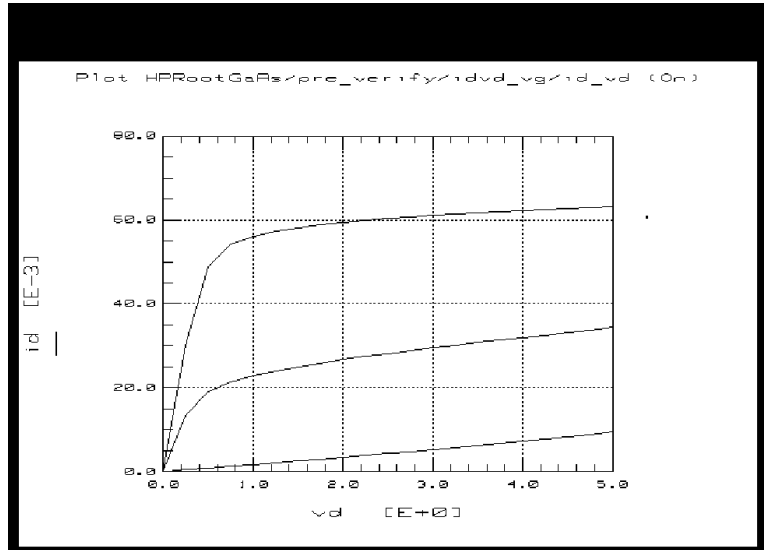
Figure 3   Example Measured pre_verify id_vd Data

4 Check the id_vd plot. The data displayed should resemble Figure 3, with the lowest curve close to the X-axis to show the pinch-off characteristics. If the data indicates the device is not operating in its normal range, go back to the inputs and change the gate voltage values. (If more than one curve is low, change vg Start to a higher value. Or if all the curves are high, change vg Start to a lower value.) Then repeat the measurement.

5 Check the ig_vd plot. The data should resemble the following figure, with smooth curves and no significant spikes present on the traces.
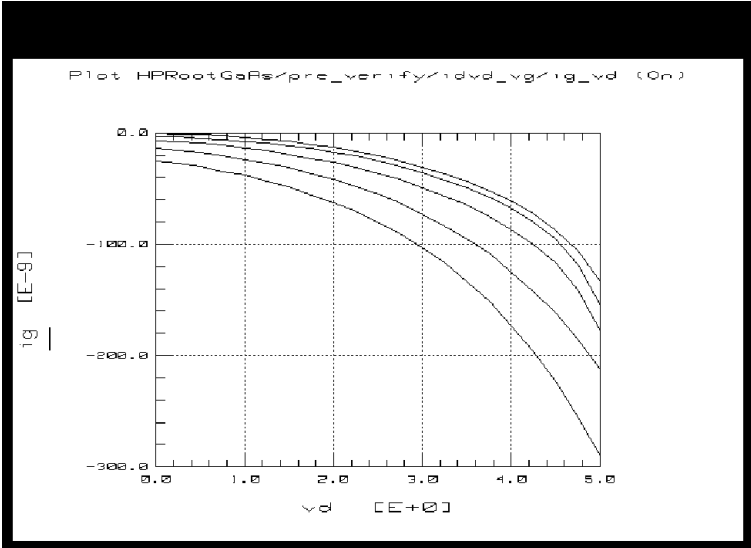
Figure 4   Example Measured pre_verify ig_vd Data

6 If the measured data looks wrong for your device, check the probe contacts, the bias connections, and the measuring instruments. Make sure the device is oriented properly, with the right ports connected to the right terminals. If none of these resolves the problem, the device may be defective. For example, if all the curves are superimposed, the gate may be shorted.

7 You can save the measured data for later comparison with data predicted by the extracted model. To do so, select **File > Save As** and a filename with the suffix *.set* (for example *dc_test.set*). Then click **OK**.

8 Close the plots.

Calibrating the Network Analyzer

It is important to calibrate the network analyzer before you perform any S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and model generation. Without a good calibration, the model *will* be inaccurate.

The Agilent Root FET measurement procedure requires two network analyzer calibration sets, a swept broadband cal and a CW cal. The broadband cal is used for the S-parameter preverification measurement. The CW cal is used for the parasitics extraction and the main data acquisition.

For a system with an Agilent 8510 network analyzer, this procedure uses the frequency list mode for both the broadband and CW calibrations. The two calibrations can be done with only one set of standards measurements, by making the CW cal a subset of the broadband cal. For a system with an Agilent 8753 or 8720 network analyzer, two separate calibrations must be performed.

Frequency list calibration procedures for an Agilent 8510-based system and an Agilent 8753-based system are provided in the next few pages. A CW calibration subset procedure for an Agilent 8510-based system is also provided.

Calibration for an Agilent 8753-Based System

Although the Agilent 8753 and 8720 have frequency list capability, IC-CAP does not support the subset calibration technique used in an Agilent 8510 system. Therefore it is necessary to perform two separate calibrations for an Agilent 8753- or 8720-based system: one frequency list cal and one CW cal. A separate set of standards measurements must be made for each calibration.

Frequency List Mode

Frequency list lets you define an arbitrary list of frequencies in one or more segments. During the measurement sweep the synthesizer is phase-locked at each frequency point, as in the stepped-sweep mode. The advantage of frequency list mode for use in a table-based model such as this is that it lets you define a smaller number of measurement points than for a step or ramp frequency cal. This reduces the total measurement time. A # of Points of 11 provides adequate measurement resolution as well as a quite fast total measurement time.

For the purpose of this model you will define two frequency segments in the list, a broadband segment and a CW segment. Set the broadband frequency list segment range at least as wide as the operating range of the device.

Set the CW frequency segment to the frequency you intend to use for the data acquisition, the same frequency you set in the init_parameters macro at the beginning of this procedure. The frequency should be above 200 MHz to eliminate the effects of test set coupler roll-off at low frequencies, and low enough to eliminate bond inductances and other parasitic inductance considerations.

Based on the broadband cal, use a calibrated measurement point equal to the start frequency plus an integer multiple of the step size: an example frequency might be 600 MHz.

$$200 \text{ MHz (start freq)} + 1 \cdot 400 \text{ MHz (step size)} = 600 \text{ MHz}$$

Frequency List Calibration for an Agilent 8510-Based System

If you are using an Agilent 8510-based system, follow the steps below.

1 On the network analyzer press PRESET.

2 To define a linear frequency list, on the network analyzer press STIMULUS > MENU > MORE > EDIT LIST > ADD. A default frequency list segment is presented.

3 To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such

as M/μ) at the right of the keypad. Similarly, press STOP and set the stop frequency.

NOTE Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys in the STIMULUS area of the front panel.

4 Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range. Keep in mind that the fewer the number of points, the less time the measurements take.

5 Press DONE to return to the frequency list menu. The segment you just defined is listed on the screen.

6 Press DONE to return to the main stimulus menu. Press FREQUENCY LIST to activate the frequency list mode.

7 Press PRIOR MENU > MORE > CONTINUAL to set a continual stimulus sweep.

8 Press PRIOR MENU > POWER MENU, and set the desired power level. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If the power level at the sampler goes above ≈−8 dBm, an **IF OVERLOAD** error message is displayed and you will need to reduce the source output power. The default network analyzer power level is 0 dBm.

9 For a device with power drop-off at higher frequencies, you may wish to set a power slope using the stimulus menus. An appropriate power slope would be in the region of 2-3 dB/GHz.

10 Press RESPONSE > MENU > AVERAGING ON/restart, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. Although the default is 256, you may wish to use an averaging factor as low as 16 to reduce overall measurement time.

11 If the cal kit constants for your calibration kit are not loaded into the network analyzer, load them from disc now by

pressing DISC > LOAD and specifying the data type and file name.

12 If you wish to modify one of the internal cal kit definitions (see Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"), do so now.

13 Press CAL > CAL1 or CAL2 (depending on your cal kit), and perform a two-port calibration, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete.

14 At the end of the calibration sequence, the cal set numbers are listed in the softkey menu. Press CAL SET #1 to store the calibration in cal set #1. (Or use another available cal set.)

CW Calibration Subset for an Agilent 8510-Based System

1 To create a CW frequency subset of the broadband frequency list calibration, press CAL > CORRECTION ON > CAL SET 1 (this turns on the broadband cal you just completed), then MORE > MODIFY CAL SET > FREQUENCY SUBSET. Use the SUBSET: START and SUBSET: STOP softkeys in the SUBSET menu (NOT the front panel keys) to set both start and stop to the same frequency.

2 Press CREATE & SAVE, and store this calibration in cal set #2 or another available cal set.

3 Note the numbers of the cal sets where you save the calibrations: you will need to list one of these numbers as part of the instrument state for *each* of the network analyzer measurements.

NOTE	For more detailed information, refer to the Agilent 8510 *Operating and Programming Manual*.

Frequency List Calibration for an Agilent 8753-Based System

If you are using an Agilent 8753-based system, follow the steps below.

1 On the network analyzer, press LOCAL to gain front panel control. Press PRESET to return to a known standard state.

2 If you are using a system with the 6 GHz receiver option and you wish to measure in the 3 GHz to 6 GHz range, press SYSTEM > FREQ RANGE 3GHz6GHz.

3 To define a linear frequency list, on the network analyzer press MENU > SWEEP TYPE MENU > EDIT LIST > CLEAR LIST > YES > ADD. A default frequency list segment is presented.

4 To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as M/m) at the right side of the keypad. Similarly, press STOP and set the stop frequency.

NOTE   Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys on the front panel.

5 Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range, then press the x1 terminator key. Keep in mind that the fewer the number of points, the less time the measurements take.

6 Press DONE to return to the edit list menu. The segment you just defined is listed on the screen.

7 Press DONE to return to the sweep type menu. Press LIST FREQ > SINGLE SEG SWEEP to activate the frequency list mode and set the analyzer to sweep the segment you defined.

8 Press RETURN > RETURN > TRIGGER MENU > CONTINUOUS to set a continuous stimulus sweep.

9 Press MENU > POWER and set the desired power level, then press the x1 terminator key. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the

device, and set a power level that will not saturate the input port samplers of the analyzer. If a receiver input is overloaded (>+14 dBm), the analyzer automatically reduces the output power of the source to −85 dBm and displays the error message **OVERLOAD ON INPUT (R**, **A**, **B) POWER REDUCED**. In addition, the annotation P↓ appears in the left margin of the display to indicate that the power trip function has been activated. When this occurs, reset the power to a lower level, then toggle the SOURCE/PWR on/off softkey to switch on the power again.

10 Press AVG > AVERAGING FACTOR, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. A good default averaging factor is 256. To speed your measurements, you may find it convenient to set an averaging factor as low as 16. Press AVERAGING ON.

11 You can further reduce the noise floor by reducing the receiver input bandwidth. Press IF BW in the average menu, and enter one of the following allowed values in Hz: 3000, 1000, 300, 100, 30, or 10. A tenfold reduction in IF bandwidth lowers the measurement noise floor by about 10 dB; however, the sweep time may be slower. For more information on averaging and the different trace noise reduction techniques, refer to the Agilent 8753 operating manual.

12 Press CAL > CAL KIT, and select the appropriate default or user-defined cal kit for your calibration devices.

13 If you wish to modify an internal calibration kit definition (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"), do so now.

14 Press CAL > CALIBRATE MENU, and perform the calibration of your choice at the analyzer front panel, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete. A full two-port cal provides the greatest accuracy. A TRL* or LRM* cal is an appropriate alternative for in-fixture measurements (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"). Omit isolation cal. Press DONE, and save the cal in the register number specified in the instrument options table.

15 Detailed procedures for measurements of calibration standards are provided in the Agilent 8753 operating manual.

CW Calibration for an Agilent 8753-Based System

Refer to Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" for a CW calibration procedure that can be used for a system with an Agilent 8753 or 8720 network analyzer.

Preverifying the Device S-Parameter (AC) Data

This procedure uses the s_vgvdf setup to measure S-parameters across a calibrated frequency range, with the drain bias voltage swept from 0V to the upper limit of the normal device operating range, and several values of gate bias.

The procedure can be used as a quick check that the device performance is good at higher frequencies, before you do a complete data acquisition and model generation. It is not necessary to do the preverification every time you generate a model, especially if you are using data sheet values.

NOTE   The frequency range and number of points set for the Input must correspond with the swept broadband calibration of the network analyzer, or the calibration will be invalid.

Defining the Instrument States

Select the **Instrument Options** tab and follow the steps described in the section "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system), with the exceptions noted below.

Because you used a frequency list cal rather than a swept frequency cal, note these differences:

- Be sure the instrument states you set for the network analyzer correspond with the values you set in your calibration.

- Set Use Fast CW to No.
- Be sure to set the averaging factor the same as in your calibration.
- Set Cal Type (SHN) to H for hardware.
- Set Cal Set No to the cal set where you stored your broadband frequency list calibration.
- Set Use Linear List to Yes.

Setting the s_vgvdf Inputs

This procedure defines the input signals from both the DC source/monitor and the network analyzer for this measurement.

Since this measurement is intended only to preverify the device, the values can be quite conservative and need not stress the device to the boundaries of its performance.

1 Select the **s_vgvdf** setup.

2 Set the vd Compliance no greater than the maximum breakdown current value for the device.

3 Set the vg Compliance no greater than the maximum allowable gate current value.

4 Set the vd range with a Start value of 0.0, and a Stop value ≤ the upper limit of the device's normal operating range. For # of Points, 3 is sufficient to display the data while giving fast results.

5 Set the vg range (Start, Stop, and # of Points) to measure at about three different bias points, in a range from approximately Vgs=Vpinchoff to Vgs=0V.

6 Set the freq Mode to F, Sweep Type to LIN, and Sweep Order to 1.

7 Set the freq Start and Stop values to correspond with the frequency range of the broadband frequency list calibration you performed.

8 Set the # of Points to correspond with the calibration.

NOTE   If you set a MHz value, use *MEG* not *M*, because IC-CAP is not case-sensitive and interprets both *m* and *M* as milli.

Measuring and Plotting

NOTE   Before performing the S-parameter measurements, manually take the network analyzer out of hold sweep mode. Press STIMULUS > MENU > MORE > CONTINUAL on an Agilent 8510, or MENU > TRIGGER MENU > CONTINUOUS on an Agilent 8753, to initiate continuous sweep.

1 Select the **Plots** tab.

2 Select **Display All**. Two S-parameter plots are displayed.

3 Select **Measure/Simulate > Measure**.

4 Check the S-parameter plots. The data displayed should be typical for your device.

   If the results are unexpected, recheck the values you set in the inputs and instrument options. In particular, be sure the instrument states and input stimulus signals match the values set in the network analyzer calibration.

   If you change the inputs or instrument states, repeat the measurement. If the data still looks abnormal, the device may be defective and may need to be replaced with an acceptable one.

5 To save the measured data if you wish, select **File > Save As**, type in an appropriate filename (for example *spar_test*), and click **OK**.

If you are satisfied with the measured data for your device, continue with the procedures.

Measuring and Extracting the Ideality Factor

It is necessary to extract the ideality factor (n) so that the model generator can use it to calculate and de-embed the gate diode resistance, Rg, where n is the coefficient that multiplies $V_{thermal}$ in the ideal diode equation. The diode resistance is approximated by the equation:

$$R_{diode} \approx \frac{nV_{th}}{I_g}$$

where n is the ideality factor, V_{th} is $V_{thermal}$ (0.025V at room temperature), and Ig is the gate current.

In this setup, the FET is configured with both the gate-source and gate-drain diode junctions strongly forward biased and intrinsic Vds=0. The drain current is set to exactly one-half the gate current, to force Is=Id, and the drain current is synchronized to the gate current. This makes the intrinsic Vds equal to zero, removing the channel resistance, and the device can thus be modeled as a T-network. The gate current is swept from zero to maximum, and Ig is measured as a function of Vg.

The inputs you set in the following procedure configure the FET as shown.
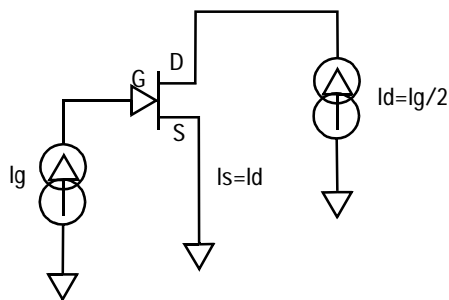
Figure 5   FET Bias Configuration for Ideality Measurement

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the r_series and idvd_vg measurements. Use the same settings.

Setting the ideality Inputs

Most of the input parameters for this measurement are factory preset and should not be changed.

1 Select the **ideality** setup.

2 The voltage Compliance can be set quite high for both ig and id, because the current will be forced only up to the maximum gate current limits set in the init_parameters macro.

3 The ig Sweep Type must be set to `LOG` scale, to allow measurement of the inherently exponential distribution of points.

4 The id Sweep Type must be set to `SYNC`, to synchronize the drain current to the gate current.

5 The purpose of the Start and Stop settings for ig is to force current from 0 amps to maximum, exercising the diode junction to include the region of its exponential behavior. Stop must be set to `IFwd`, which is the forward conducting gate current compliance. Start must be set equal to IFwd divided by some large number between 1000 and 100000.

   When the ig Sweep Type is set to LOG, the input table automatically adds the fields Dec or Oct and Total Pts in place of Step Size.

6 Dec or Oct refers to a decade or octave log scale for the plotted data. The correct setting for this measurement is `Dec`.

7 To set the number of points, you can enter either a # of Points per decade, or a Total Pts across the sweep.

   When the id Sweep Type is set to SYNC, the input table automatically adds fields for Ratio, Offset, and Master Sweep.

8 The id Ratio must be set to `-500.0m` to make the drain current exactly -1/2 the gate current.

9 Offset must be set to `0.000`.

10 The id Master Sweep must be set to `ig` to synchronize Id to Ig.

11 The Outputs are factory set, and should not be changed.

Measuring, Extracting, and Interpreting the Results

This procedure makes the measurement and computes the ideality factor from the results. The procedure explains how to check the measured data and computed ideality, and override the computed value if appropriate.

1 Select the **Plots** tab.

2 Select the **n_vs_vg** plot, and **Display Plot**.

3 Select **Measure/Simulate > Measure**. Check the plot of n (ideality) as a function of Vg. The displayed trace should resemble the following figure. Note that n is computed as the minimum value along the linear region of the logarithmically displayed curve, and should be a value between 1 and 2.
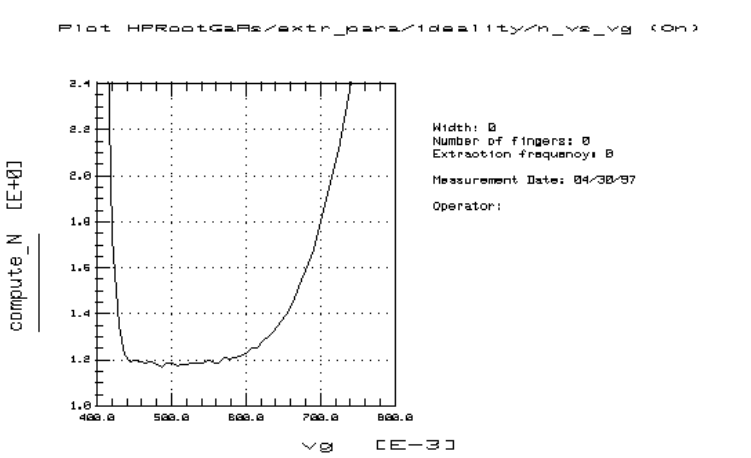
Figure 6   Example n_vs_vg Ideality Plot

4 Select the **Extract/Optimize** tab.

5 Select **compute_N > Execute**. The transform calculates the ideality factor from the measured data, and puts the value

into the extr_para DUT variable table. You can view the DUT variable table by selecting extr_para > DUT Variables.

6 You can check the ideality factor in the variable table against a measured value on the trace. Put the mouse pointer at a selected point on the linear portion of the trace, click the middle mouse button, and a readout of the Y-axis value at that point will be displayed. (Note that the linear region of the measurement is the region where the R_{diode} equation given under "Measuring and Extracting the Ideality Factor" on page 39 applies.)

   If the n_vs_vg plot looks significantly different from the illustration, it may be necessary to modify the transform or the computed value, as explained in the following steps.

7 If noise is present on the trace below the linear region of the curve, n may have been computed as the minimum value of the noise. In this case, go back to Inputs and change the ig Start value from IFwd/10000 to IFwd/1000 to start the measurement at a value of current above the noise level. Execute the compute_N transform again, observing the change in the plot and in the ideality value in the variable table.

8 If no measured linear region is shown on the trace, a good ideality factor cannot be computed from it. In this case, you will need to manually enter a value of 1 or 0 for ideality. (In the R_{diode} equation, the nV_{th} term is relatively small compared with Ig. If a good ideality factor cannot be computed, it is best to eliminate the contribution of the nV_{th} term by using an n value of 0.)

9 To change the ideality factor in the variable table, move the mouse pointer over the value and click left to highlight it, then type in the new value.

10 The illustration below is a typical ig_vs_vg Plot.
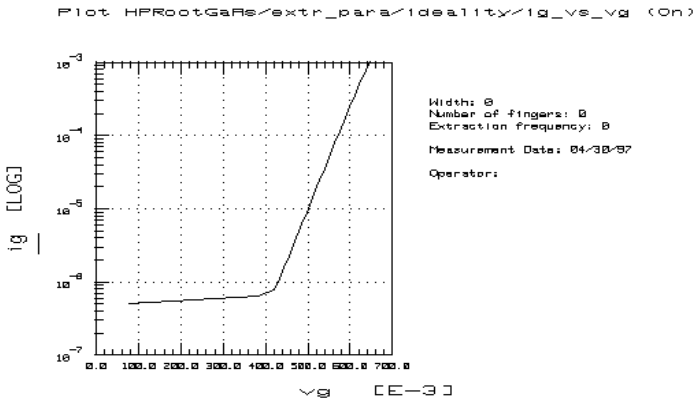
Figure 7   Example ig_vs_vg Ideality Plot

Measuring and Extracting Parasitic R and L Values

This procedure performs an S-parameter measurement at a single CW frequency, with the FET in the same configuration as for the ideality measurement. Drain current is set to one half gate current, to force Is = Id = −1/2 Ig. This measurement is performed at the same CW frequency as the main data acquisition, using the same network analyzer CW calibration.
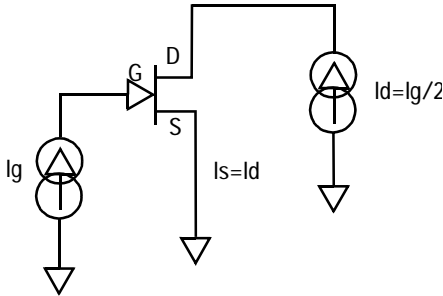
Figure 8   FET Bias Configuration for parasitics Measurement

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Instrument States for Swept Frequency Measurements" on page 624 (Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Set the Cal Set No to the cal set or register where you saved your broadband calibration. The procedures in this manual use cal set #1.

Setting the parasitics Inputs

Most of the input values for this measurement are factory preset and should not be changed.

1 Select the **parasitics** setup.

2 Set the voltage Compliance for both *ig* and *id* to any reasonable values. The values can be quite high because the current is limited by the forward conducting gate current value you set in the init_parameters macro.

3 The Sweep Type is set to CON (constant) for both *ig* and *id*.

4 The ig Value must be set to IFwd, (the forward conducting gate current value you set in the init_parameters macro).

5 The id Value must be set to -IFwd/2, to force Is = Id.

6 Under freq, the Mode must be set to F and the Sweep Type to LIN (linear).

7 Set the freq Start, Stop, and # of Points to correspond with the swept frequency calibration you performed.

Measuring and Extracting

1 In the parasitics setup select **Measure**.

2 When the measurement is complete, select the **Extract/Optimize** tab.

3 Select **extract_parasitics > Execute**. The transform calculates the parasitic R and L values from the measured data.

Setting Up the Main Data Acquisition

This sets up the values for the main measurement procedure, which takes the data from which the model is generated. The procedure measures S-parameters at a calibrated CW frequency, across a range of drain and gate bias voltages, with the gate forward biased and intrinsic Vds = 0. It takes adaptive measurements, densely spaced in the most nonlinear regions such as at the knees of the I-V curves and at the onset of breakdown, and less densely spaced in the linear saturation region. The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

Some knowledge of the device characteristics is needed in order to enter valid parameters. The data you measured in the DC and S-parameter preverifications may be useful in determining appropriate values for your device. Do not exceed maximum allowable values. Perform the steps in the following paragraphs.

Once you have initiated the transform, the data acquisition procedure takes approximately 1 to 2 hours to run before your interaction is needed again. The data acquisition rate is approximately 700 points per hour.

Defining the Instrument States

The instrument states for the main data acquisition are set in the dc_bias and ac_bias setups.

CAUTION   Do not change the Inputs in the dc_bias, ac_bias, or create_mdl setups. Their function is not the same as in the other setups. They contain factory-set information essential to the program, and they *must not be changed*.

1   Select the **dc_bias** setup.

2   Select the **Instrument Options** tab to display the DC source/monitor instrument states.

3   Set the Agilent 4142 instrument state the same as for the DC preverification. Refer to the appropriate chapter as follows:

   • Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

   • Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

4   Set Use User Sweep to No.

5   Select the **ac_bias** setup.

6   Set the instrument states the same as for the parasitics measurement, using the same network analyzer CW frequency calibration.

7   Set Use User Sweep to Yes and Use Linear List to Yes.

8   Set the Cal Set No to the cal set where you stored your CW calibration.

Setting the data_acquisition Variables

You set the values that define the measurement and model generation in the data_acquisition and model_generator transforms in the create_mdl setup. Based on your information, IC-CAP characterizes the device within a subset range of the bias plane (Vd-Vg plane). This range is typically larger than the normal safe range for long-term DC device operation specified in the data sheet. In other words, the device is briefly pushed beyond its long-term normal DC operating range. This is required by the model for simulating large-signal AC and

transient pulsed operation, where the instantaneous voltages can temporarily exceed the boundaries of long-term safe DC operation.

1 Select the **create_mdl** setup.

2 Select the **Extract/Optimize** tab, and the create_mdl transforms are listed.

3 Select **data_acquisition** to display the table of variables used in the data acquisition transform (Figure 9).
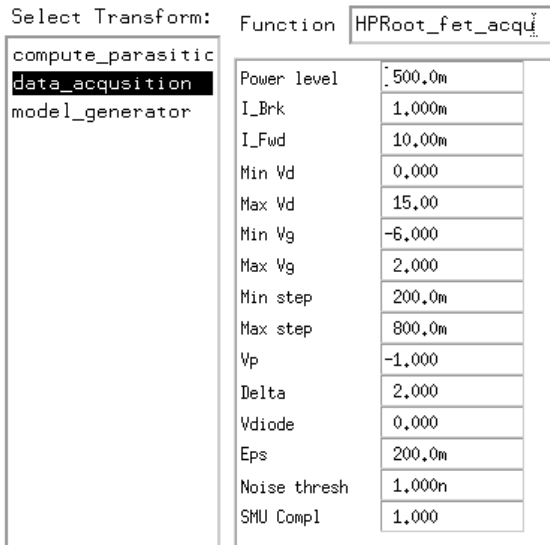
Figure 9   Variables in the data_acquisition Transform

4 Follow the steps below to enter variable values appropriate for your device.

5 For Power level, enter the total maximum power dissipation value. Typically this should be 2 watts per mm of gate width for devices up to 1 mm.

6 For I_Brk, enter the reverse gate current breakdown value, typically 20 mA per mm of gate width.

7 For I_Fwd, enter the forward conducting gate current, typically 200 mA per mm of gate width.

8   The next four voltage values define hard limits to the measurement domain. The voltage values are set well beyond the normal device operating range to explore the device's boundary response. If the values are set too conservatively, data will not be measured in the nonlinear areas of breakdown and maximum forward conduction. If this occurs in the measurement, return to this table and change the voltage values. Reduce the minimum values and increase the maximum values.

9   Min Vd should typically be slightly negative, to allow data to be taken slightly beyond the bounds of the model so that it possible to properly extrapolate beyond the data.

10   Max Vd should typically be slightly higher than any anticipated breakdown voltage.

11   Min Vg should typically be several volts lower than Vpinchoff.

12   Max Vg should typically be greater than the voltage required to achieve the maximum fundamental gate current.

13   For Min step, enter a value to define the step size between densely-spaced measurements.

14   For Max step, enter a value to define the measurement step size where you expect the data to be relatively linear and you do not need densely-spaced measurements.

15   For Vp, enter the gate pinch-off voltage where drain voltage = 0V.

16   Delta defines a voltage range below Vpinchoff, where a high density of Vgs measurements is needed due to the nonlinear decrease in drain current at pinch-off. A typical value for Delta is 2 volts.

17   Vdiode defines a voltage above which high-density measurements are needed, because of nonlinearities caused by the increase of gate current in diode junction forward conduction mode. A typical value for Vdiode is between 0V and the diode junction turn-on voltage.

18   Eps (Epsilon) is used to predict the next point to be measured, and thus control automatic data acquisition step size. Increasing the value of epsilon decreases the sensitivity of the automatic data acquisition.

19 For Noise thresh, enter a current threshold value for both drain current and gate current below which you do not need closely-spaced measurements. The automatic step-size algorithm is not used below the Noise thresh value, where measured currents are sampled at the maximum step size.

Do not make Noise thresh so small that it measures numerous points below pinch-off, wasting measurement time and storage capacity. On the other hand, do not make it too large. A typical value is 5.0×10^{-7} amps per mm.

20 For SMU Compl, enter the drain SMU maximum current compliance value in amps. This is the maximum output current capability of the Agilent 4142B plug-in source/monitor unit (SMU) supplying the drain terminal. In the example procedure, this is VD. For the Agilent 41420A high-power SMU, the compliance is 1A; for the Agilent 41421B medium-power SMU, the compliance is 100 mA.

CAUTION   Also be aware of the current ratings of your bias networks, and do not set current values beyond their specified maximum limits. It is quite possible to destroy the bias networks, and the probes, by applying current values beyond the specified limits. (The standard bias networks for the Agilent 85122A/85123A parameter extraction systems are rated at 0.5 amp.)

21 You can save the values in the data_acquisition transform. Choose **File > Save As** and select **Transform** as the File Type. Type an appropriate filename, and click **OK**. The file is saved with the suffix *.xfm* for transform.

Performing the Data Acquisition Measurement

This procedure performs the measurement you defined. Once you initiate the transform, the data acquisition takes approximately 1 to 2 hours to run before your interaction is needed again. The data acquisition rate is approximately 700

points per hour. The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

1 In the create_mdl setup select the **Extract/Optimize** tab.

2 Select the **data_acquisition transform**, and **Execute**.

CAUTION   Do not perform data acquisition more than once. This will write over the existing data, and will use more time and memory.

3 If you do attempt to perform data acquisition after the measurement has already been done, a dialog box will alert you that a measurement file exists. You then have several choices: If you do not need the data in the current file you can click **OK** or press Enter to overwrite it. Or you can abort your new measurement by selecting Cancel. Or if you wish to save the measured data and perform another data acquisition, go to a UNIX window, change directories if necessary, find the file *Measured.data*, and rename it with another appropriate name, then continue with the data acquisition.

Ignore this message if it occurs: **CAUTION: CALSET X STATE MAY BE INVALID**.

4 During the data acquisition, a running message in the Status window indicates the approximate percentage of completion.

5 When the process is complete, a message similar to the following is displayed:

```
INITIAL Extraction Results

Max_num = 66.00

Max_row = 49.00
```

The numbers are part of the internal function of the program, and irrelevant in user interaction with the program. They are associated with the maximum number of points for a given Vgs, and the number of Vgs steps.

Plotting the Measured Data

The plots in the create_mdl setup provide graphical outputs of the data acquisition and model generation. For hardcopy output, black and white can be reversed for faster printing and easier reading on paper, as explained here.

If you wish to plot data from a previously stored data set, you will need to perform the steps explained under "To Plot an Earlier Data Acquisition," on the next page. Otherwise, follow these steps.

Viewing and Printing the Plots

1 Select the **plots** tab.

2 Select the plot of your choice, and **Display Plot**.

3 You can augment the annotation of a plot by specifying header and footer labels you wish to print with the data. However, it is *essential* not to change the Y data input fields.

4 To change a plot's annotation, go to the Header or Footer lines in the plot definition. You can type a label of your choice in the header or footer input field (right side of the definition table). Then select **Options > Update Annotation**, and your label is added to the plot.

NOTE   Changing the Y Data n input fields can cause anomalies in the measurement and modeling results.

---

5 If you wish to exchange the black and white in a plot for better hardcopy quality, select **Options** from the plot menu, and **Exchange Black_White** from the pull-down menu. You can also change from colored traces to all black by selecting **Options > Color**. This is particularly useful if one of the traces is a light color such as yellow, which may not show well against a white background.

6 To get a hardcopy of a plot, from the plot menu select **File > Print Setup**. Enter the name of a printer and make any other changes to the printer/plotter setup you wish, then click **OK**.

7 If you have your printer setup completed, select **File > Print**.

To Plot an Earlier Data Acquisition

If you wish to view data from a previously stored Agilent
RootFET data acquisition, you will need to perform these
additional steps to bring the data into this model file and
display the plots.

1 From the IC-CAP main menu select **File > Change Directory**.
Type in the pathname of the file where your data is stored,
and click **OK**.

2 Select the **create_mdl** setup.

3 From the Plots tab, select the plot of your choice, and **Display
Plot**. The displayed plot window will be blank and an error
message will be displayed. *This is not a problem.* Click **OK** in
the error message window and the error will be erased.

4 Now select **Display Plot** again, and the plotted data will be
properly displayed. (The first time you selected Display Plot,
an internal function re-initialized the data from the changed
directory.)

The following figures illustrate the data measured by the
data_acquisition transform for an example device.

• Id_vd plots the DC measurement of drain current Id as a
function of Vds for different values of Vgs (Figure 10).
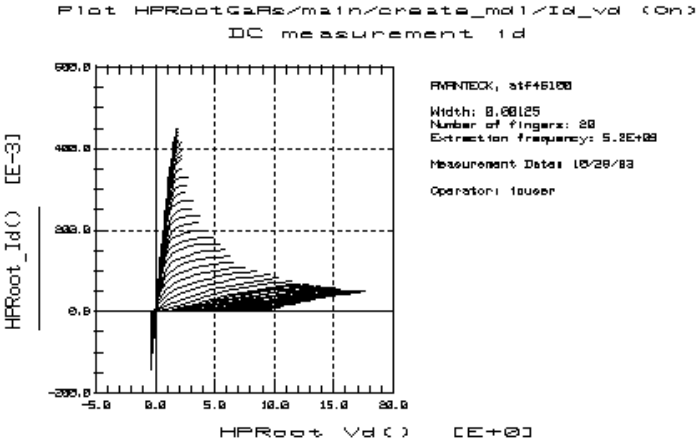
Figure 10   Example Id_vd Data

If you are not satisfied with the bias range of the measurement, make the necessary changes to the values of Min Vd, Max Vd, Min Vg, and Max Vg.

• Ig_vd plots the DC measurement of gate current Ig as a function of Vds for different values of Vgs. The following figure illustrates an example of this data. The concentration of traces around Vgstart may be unfamiliar but is correct for the very strongly forward-biased conditions.
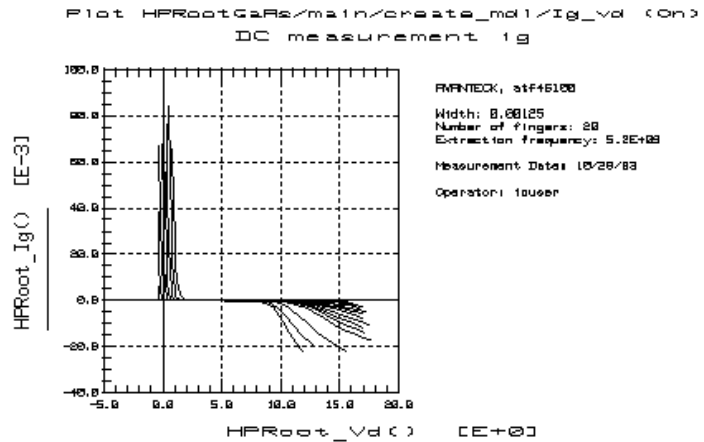
Figure 11   Example Ig_vd Data

- Y11i plots the imaginary data for Y11.

- Y12i plots the imaginary data for Y12.

- Y21r plots the real data for Y21.

- Y22r plots the real data for Y22.

- Vd_Vg plots the distribution of test data in the Vgs-Vds plane. It shows how the data is taken adaptively depending on the device nonlinearities. More test data is taken where the device characteristics change rapidly such as at the knees of the I-V curves and the onset of breakdown, and less in the linear saturation region. The following figure illustrates an example of this data.
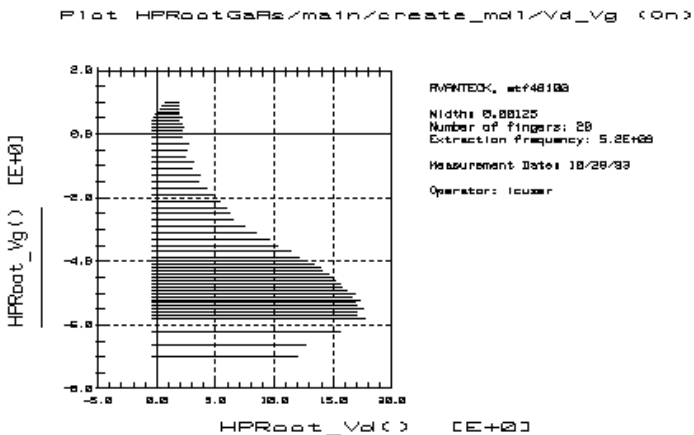
Figure 12   Example Vd_Vg Data

If you are satisfied with the measured data for your device, proceed to the model generation. Be sure the limits of the measurement domain are adequate: the domain should extend beyond the normal operating range of the device. If the measured data does not show the limits of device behavior, you may wish to redefine the voltage ranges you set in the init_parameters macro, using values that are less conservative.

Generating the Model

This procedure calculates the model parameters from the measured data, and generates a corresponding model for the device. The model nonlinear functions are calculated by contour integration, therefore you must provide the program with a starting point for the contour integration.

If you are generating a model from a previously stored data set, perform the steps listed later in this procedure under "To Generate a Model Using Data from an Earlier Measurement."

Computing the Parasitics for the Model Generation

An earlier part of the procedure calculated the intrinsic parasitic resistances of the device and entered them in the file Para.data. Now this procedure updates the data and gives you the opportunity to override the calculated values and enter values of your own if you believe they are more appropriate.

1 Select the **main** DUT, and the **DUT Variables** tab. A variable table is displayed, showing current values for the intrinsic parasitic resistances and inductances, as well as other variable values. The figure on the next page illustrates the DUT variables table.

2 If you have reason to change any of the parasitic values, you can change them in this variable table. For example, if you have calculated the parasitic values by another method, such as from material measurements of by using a different algorithm, you may prefer to use those values.

3 Or based on the model generation results plotted later in the procedure, you may wish to return to this step and modify values, then perform the transforms again.

4 To change a parasitic value in the variable table, move the mouse pointer over the current entry and click left to highlight it, then type in a new value.

5 Do not change any of the other variable values in the table besides the R and L values. The other variables listed are internal functions necessary to the program, and can be ignored. *Do not change them.*

6 If you make any changes to the parasitic values in the table, *do not* execute the compute_parasitic transform.

Parameters	DUT Variables
Name	Value
Rs	0
Rd	0
Rg	0
Ls_tot	13.77p
Ld_tot	67.35p
Lg_tot	37.67p
Ls_ext	0
Ld_ext	0
Lg_ext	0
Cpds	0
Cpgs	0
Cpgd	0
Max_num	21.00
Max_row	29.00
Vdrain	8.0
Vgate	-6
Idrain	1e-9
Igate	2e-8
Id_Compliance	0
Ig_Compliance	0

Figure 13   Variables Table in the main DUT

7 If you do *not* wish to make changes to the parasitic values in the table, select the **compute_parasitic** transform, and **Execute**. You will probably observe changes in the parasitic values in the table.

> **NOTE**
>
> You can copy the variable table by selecting Detach to display it in a separate window. Resize the window and move it to a convenient place away from the central work area.

Setting the Starting Point for Contour Integration

NOTE   For the modeled nonlinear functions to be valid, contour integration must
be done in the normal operating range of the device. Outside the range, the
results will be very different and less accurate. The further from the
operating range, the less accurate the model will be.

Use these guidelines to set the starting point of contour
integration:

1 The starting point should be set at an intersect of Vd and Vg
  that is within the saturated class A operating range of the
  device, where drain voltage is beyond the knee and less than
  breakdown, and gate voltage is between zero and Vpinchoff.

2 If you know Vpinchoff and Vbreakdown, set:

  • Vpinchoff < Vg_start < 0
    and
    0 < Vd_start < Vbreakdown

  or

  • Vg_start = Vpinchoff/2
    and
    Vd_start = Vbreakdown/2

NOTE   To generate a good model, make sure sufficient data points are available
around the Vg,Vd points chosen for the starting point of the contour
integration. Before running the model generator, look at the Id_Vd plot
under the create_mdl setup. Use data markers to see the discrete points at
which data was taken. The values you choose for the Vg,Vd values for
contour integration should be well within the *data space* over which data
was taken. Note that this will be bias value at which the best fits are
achieved for the Root Model.

Setting the model_generator Variables

1 Select the **create_mdl** setup.

2 Select the **Extract/Optimize** tab to display the list of transforms.

3 Select **model_generator** to display the table of variables used in the model generator transform (see illustration).

Select Transform:	Function	HPRoot_FET_t
compute_parasitics		
data_acqusition	vd_start	5.000
model_generator	vg_start	−500.0m
	t_dispersion	100.0n
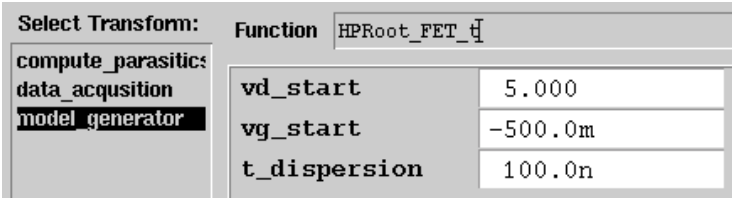
Figure 14   Variables in the model_generator Transform

4 In the variable table, click left on the vd_start value to highlight it, then type in an appropriate value for your device.

5 Similarly, set an appropriate vg_start value for your device.

6 Enter an appropriate value for t_dispersion. This is the dispersion time constant, and its default value is 1e−7 or 100 nanoseconds, 100.0n. In most cases, t_dispersion is not a significant factor although it is used in the model generation calculations. However, in a case where you wish to add time dispersion effects, you can increase the value.

Executing the Model Generator Transform

1 Select **model_generator > Execute** to start generating the model for the device under test.

2 A dialog box asks you for an MDS file name. The standard file generated by the model is *State.mds*. This file can be used with both ADS and MDS. If you want to retain this filename click **OK** or you can define a different filename.

<table>
<tr><td>NOTE</td><td>If you plan to do a verification, retain the filename *State.mds* for now, as the verification process requires this filename. You will have another opportunity to change the filename after the verification process.</td></tr>
</table>

The model generation takes a few seconds. When the model generation is complete, a message is displayed.

Generating a Model Using Data from an Earlier Measurement

If you wish to generate a model from a previously stored measurement, you will need to perform these additional steps to bring the data into this model file in the correct form.

1 From the IC-CAP main menu select **File > Change Directory**. Enter the path of the file where your data is stored, and press Return.

2 Select the **create_mdl** setup.

3 Select the **Extract/Optimize** tab.

4 Select the **compute_parasitics** transform, and **Execute**.

5 Now perform the standard procedures "Setting the Starting Point for Contour Integration" on page 58 and "Executing the Model Generator Transform" on page 59.

<table>
<tr><td>NOTE</td><td>To plot the data, note the extra steps described in "To Plot an Earlier Data Acquisition" on page 52.</td></tr>
</table>

Plotting the Generated Data

In the create_mdl setup, the Idh, Qg and Qd plots show the calculated state function data generated by the model_generator transform.

NOTE The modeled state function data will not be plotted unless the init_parameters macro has been executed (see the instructions near the beginning of this chapter). The *icplotnotes* file generated by the init_parameters macro provides the x-axis information needed to plot the modeled data. IC-CAP will display an error message if you have not executed the init_parameters macro.

To view a specific plot:

Select the plot and click **Display Plot**.

- Idh plots the high-frequency drain current Id as a function of Vds for different values of Vgs.

- Qg is the distribution of charge current under the gate.

- Qd is the distribution of charge current under the drain.

Other Outputs of the Model Generation Process

The model generation process results in a number of different output files to be used for different purposes. These include the model file for MDS interface, data on various modeled parameters, measured device data, and files for internal use by the program.

Some files have default filenames assigned by the program. Others may be files you created during the procedure to save device-specific information.

CAUTION In general, the files should *NOT* be modified, because changing them can cause problems that may invalidate the model. It is also important that you do not change the names of files created by the model generator, with the exception of *State.mds*.

The following list of files will be (or may be) created during the modeling process:

- *icplotnotes.* This is the only file you can safely edit or add to. The program appends it as a label to any graphical outputs, to identify the device. It includes a device ID string and any additional information you choose to have added to graphs when they are plotted.

- *icplotnumbers.* This is an internal file used by the program to generate a graphical interface. It is related to the Max_num and Max_row messages that appear in the parasitics variable table of the main DUT and in the UNIX window at the end of the data acquisition. While these parameters are irrelevant to user interaction with the program, they are essential to its function, and they must appear on-screen. *Do not change this file.*

- *Measured.data.* This is the raw measured S-parameter data as a function of bias and frequency, produced by the create_mdl data acquisition process.

- *Para.data.* This is the S-parameter data with forward bias current used by the program to calculate parasitic resistance and inductance values.

- *State.data.* This file is used by the program in plotting nonlinear functions such as Qg and Idh. *Do not change this file.*

- *State.mds.* This is the model file used by the Agilent RF and Microwave Design System (MDS) in a circuit simulation. It is also used in verifying the model, and needs to retain the name *State.mds* in the verification process in order to maintain the MNS simulator link. However, since every Agilent RootFET model initially has this name, you change the name to identify the device, either when you initiate the transform to generate the model (if you are not performing a verification), or at a later time as described at the end of this chapter.

- *Y.data.* This file contains the Y-parameters as a function of bias, to examine through graphical interface. *NO changes* can be made to the *Y.data* file, *except* for adding header and footer labels as explained under "Plotting the Measured Data," earlier.

- *dc_test.set* (or some other name designated by you). If you decided to save it, this is the measured DC data generated in the DC preverification procedure.

- *spar_test.set* (or some other name designated by you). If you decided to save it, this is the measured S-parameter data generated in the S-parameter (AC) preverification.

Verifying the Model DC Data

This procedure verifies the model DC data against the DC data measured in the idvd_vg setup, using the MNS simulator link.

Notes

- The simulator used must be MNS. If the SIMULATOR variable in the model variable table is set to another value, change it to mns.

- The name of the model file must be *State.mds* for the purposes of verification, to maintain the MNS simulator link.

1 Select the **idvd_vg** setup. Make sure the output Type is set to **B** for both measured and simulated data.

2 Select the **Plots** tab.

3 Select the **id_vd** plot, and **Display Plot**.

4 Under Measure/Simulate select **Simulate**.

5 Check that the simulated data (dotted line) and measured data agree.

Verifying the Model S-Parameter Data

This procedure verifies the model S-parameter data against the S-parameter data measured in the s_vgvdf setup, using the MNS simulator link.

Notes

- The simulator used must be MNS. If the SIMULATOR variable in the model variable table is set to another value, change it to mns.

- The name of the model file must be *State.mds* for the purposes of verification, to maintain the MNS simulator link.

1 Select the **s_vgvdf** setup. Make sure the output Type is set to **B**.

2 Select the **Plots** tab. Select **Display All** to display both S-parameter plots.

3 Under Measure/Simulate select **Simulate**.

4 Check that the simulated data (dotted line) and measured data agree.

Storing Data and Using it with MDS

Once you have developed your device model, you can store it in a file for future use. You can use it for circuit simulation in the Agilent RF and Microwave Design System, inserting it into circuits under design to emulate the characteristics of your device. It can be used to represent any device with the same geometry from the same process, or scaled to represent a different-sized device. It will predict the gain, power, gain compression, and harmonics performance of the device as a component in circuits you design using MDS. It will predict large-signal device operation over a range of bias points and loading conditions.

The file generated by the modeling process for interface with MDS is called *State.mds*. You need to change the name to one that will identify the device under test, since *every* Agilent Root FET model file is initially named *State.mds*.

If you chose not to perform a verification, you may already have changed the filename using the dialog box presented before the model_generator transform. If not, you can change the filename now in UNIX and store the file as follows:

Storing the Model

1 Use the UNIX directory you created at the beginning of the procedure, when you first opened the model file.

2 From the IC-CAP/Main window select **File > Save As**.

3 A dialog box is displayed. Select the File Type (*.mdl* for a complete model) by clicking the corresponding radio button.

4 Select a new filename that associates this model with the device.

5 Use the UNIX copy (cp) or move (mv) command to copy the file *State.mds* to the new filename, including the pathname of the designated directory. Click **OK**.

This completes the example Agilent Root FET measurement and model generation procedure. For an explanation of how to use IC-CAP model data in MDS, refer to Appendix D, "Using IC-CAP Data with MDS."
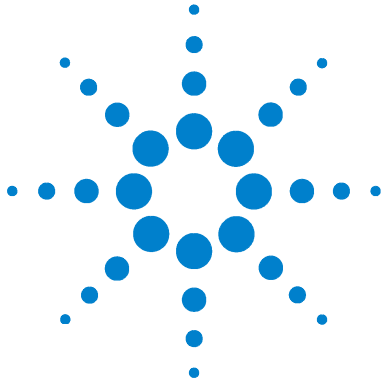
References

1   *Principles Of Nonlinear Active Device Modeling For Circuit Simulation*. David E. Root and Brian Hughes, Hewlett-Packard Company. Procedings 32nd ARFTG Conference, Tempe, AZ, pp. 3-26, 1988.

2   *Automated Data Acquisition System For FET Measurement and Its Application*. Sigi Fan, David E. Root, and Jeff Meyer, Hewlett-Packard Company. Proceedings 38th IEEE MTT ARFTG Conference, San Diego, CA, December 1991.

3   *Measurement-Based Active Device Modeling For Circuit Simulation*, European Microwave Conference, 1993.

4   *A Measurement-Based FET Model Improves CAE Accuracy*. David E. Root, Hewlett-Packard Company. Microwave Journal, September 1991.

5   *Measurement-Based Large Signal Diode Model, Automated Data Acquisition System*. MTT Digest, 1993.

6   *Analysis and Exact Solutions of Relaxation-Time Differential Equations Describing Non Quasi-Static Large Signal FET Models*. David E. Root, Hewlett-Packard Company. European Microwave Conference. Cannes, France, 1994.

7   *Experimental Evaluation of Large-Signal Modeling Assumptions Based On Vector Analysis of Bias-Dependent S-Parameter Data from MESFETs and HEMTs*. David E. Root and Sigi Fan, Hewlett-Packard. IEEE MTT-S Digest, 1992.

8   *Measurement-Based Active Device Modeling For Circuit Simulation*. David E. Root, Hewlett-Packard Company. European Microwave Conference, Madrid, Spain, 1993.

9   *Technology-Independent Large-Signal FET Models: A Measurement-Based Approach to Active Device Modeling*. David E. Root, Sigi Fan, and Jeff Meyer, Hewlett-Packard Company. 15th ARMMS Conference, 1991.

10 *Accurate Large-Signal GaAs MESFET Modeling for a Power MMIC Amplifier Design*. J. M. Dortu and J. E. Muller, Siemens AG, Munich, Germany. Microwave Journal, April 1993.

11 *The Extraction Of Terminal Charges From Two-Dimensional Device Simulations Of MOS Transistors.* E. J. Predeergast and P. Lloyd, AT&T Bell Laboratories, Allentown, PA. COMPEL - The International Journal for Computation and Mathematics in Electrical and Electronic Engineering, Vol. 6, No. 2, 107-114. Boole Press Limited, 1987.

12 *Direct Extraction Of GaAs Mesfet Intrinsic Element And Parasitic Inductance Values.* Eric Arnold, Michael Golio, Monte Miller, and Bill Beckwith, Motorola Inc., Strategic Electronic Division, Chandler, AZ. IEEE MTT-S Digest, 1990.

13 *A New Method for Determining the FET Small-Signal Equivalent Circuit.* Gilles Dambrine, Alain Cappy, Frederic Heliodore, and Edouard Playez. IEEE Transactions on Microwave Theory and Techniques, Vol. 36, No. 7, July 1998.

(Additional references are listed in the bibliography at the end of some of these publications.)

1 Agilent Root FET Model Generator

2

High-Frequency FET (Curtice) Models

This chapter documents the Agilent 85192A high-frequency FET modeling software which includes three models:

- Curtice quadratic
- Curtice cubic
- Statz (Raytheon)

While these models are well known for use with DC and low-frequency devices, the extraction methodology used here is specific to high-frequency devices. The measurement and extraction software and procedures have been developed to ensure accurate modeling results that are constant, rather than a function of frequency.

The example procedures here focus on the Curtice cubic model. Some differences between the Curtice cubic and Curtice quadratic models are documented in the IC-CAP *Reference* manual.

The software performs a series of DC and S-parameter measurements, based on predefined measurement configurations and on variables you define during the procedures. The measured values are then used to extract individual device parameters, through software conversion of the S-parameters to admittance or impedance parameters. The resulting model can be used to simulate the performance of any similar device in a circuit under design. Its data files can be read directly into datasets in the Agilent RF and Microwave

Agilent Technologies

Design System (MDS). The model can also be used as a standard for quantitative performance measures in process and material control.

Getting Started in IC-CAP

The procedures in this chapter assume the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"
  - Wiltron 360-based system—Refer to your Wiltron documentation
- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.
- The IC-CAP software has been properly installed on the computer.
- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename a medium-power SMU connected to the device gate as VG

Rename a high-power SMU connected to the device drain as VD

Rename the GNDU unit (which must be connected to the source) as GND.

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on launching the IC-CAP software and opening the model file you need. The Curtice FET model file is under:

/examples/model_files/mesfet/CGaashf.mdl

Then return to this chapter and continue with the procedures here.

Before you continue with the IC-CAP procedures, make a new directory in an appropriate location in the UNIX structure, using the mkdir command, to store the data and model files that will be generated. Give the directory a name that will associate it with the devices you are going to model. Use the cd command to change to the new directory in UNIX.

Using the Curtice Cubic Model

This chapter provides example procedures that take you step-by-step through a FET measurement and parameter extraction. The procedures are arranged in the most logical sequence to allow earlier measured data or extracted parameters to be used as a basis for later measurements or extractions.

The device used in this example was a GaAs MESFET, and its data sheet was used for information on device characteristics and compliance values. Values you use in your own parameter extraction procedures will depend on the particular characteristics of your device: use its data sheet (if available) as a reference.

This example is provided as a starting point. It is a set of straightforward procedures that use only a subset of IC-CAP's total capabilities. The complete procedures documented give all the steps necessary to set up and perform the measurements and extract device parameters from the resulting data. Once the measurement setups have been defined, they can be used for measurement and extraction of any device of the same type and geometry. The modeling procedure then becomes more simple, and can be accomplished in a few steps.

The procedures use a series of different setups to measure current or voltage vs bias under different bias conditions. The purpose of the different measurement setups is to decouple the model equations as much as possible and effectively isolate the individual FET parameters.

NOTE	Although you will modify the variable values in factory default files during this procedure, do not under any circumstances change the *names* of any variables or setups. These names are used by the software in the parameter extraction process.

NOTE	These example procedures assume that in-fixture measurements are made with the FET under test mounted in the fixture in a common-source configuration.

The procedures in this chapter include detailed instructions for the following:

- Setting the input parameters for each measurement.
- Making the DC and S-parameter measurements: qualifying the results.
- Extracting the individual model parameters.
- Verifying the extraction results against the measured data.
- Storing the model and individual parameters.

Where necessary, other chapters are cross-referenced at appropriate points in the procedures, so that all system hardware setups and calibrations are performed at the right times.

NOTE Measurement parameters and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct method or results.

The procedures start with example values for IC-CAP measurement inputs. They provide guidelines on setting the values to measure your device. However, the values you use will be based on your knowledge of your device and on information provided in its data sheet as well as on the guidelines provided here: they may or may not correspond with the examples. In some cases, it may be necessary to set the input values, make a measurement, and then reset the inputs based on the measurement results. The procedures give suggestions on judging the applicability of the input values and measured data.

The illustrations of plotted data are provided as visual examples of possible measurement results. They are not intended to represent a sequential measurement of one device, nor do they necessarily correspond with the example measurement inputs described in the procedures. The plots can be used to check the reasonableness of your own measured data, which should appear similar in shape but not necessarily coincident in values.

In addition, the IC-CAP software itself includes measured data for some of the setups. This data is from the measurement of a single device, and shows another example of possible results. Many, though not all, of the input settings for this measurement correspond with the example settings. In looking for information on preferred values, or on how the measured data should look, refer to the manual as a guide.

The DUTs and Setups

The DUT/Setup panel is the central access point for the measurement and extraction procedures. The DUTs (for example DC), indicated by the inverted triangles, are groupings of similar measurement setups used to extract related model parameters. The setups contain the information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.
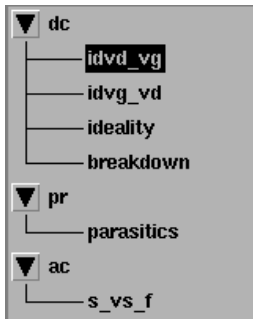
Figure 15   The CGaashf DUT/Setup Panel

Making the DC Measurements

The DC measurement procedures measure DC current or voltage vs bias under different bias conditions. Setting up the measurement conditions involves two main processes for each measurement setup: defining the instrument state for the DC source/monitor, and setting the input values for the device measurement.

Leave the network analyzer disconnected from the bias networks while you perform the DC measurements, with the RF IN connectors of the bias networks terminated to prevent possible bias oscillation.

Defining the Instrument States

For each of the measurement setups, it is necessary to define the instrument states (options) for the measurement instruments that will be used in that setup. The instrument states need to be defined individually for each setup. Different setups use different measurement instruments. However, with minor exceptions that will be explained, the instrument states for the same measurement type must be used for all the setups that use that measurement type.

1 After you have selected a setup, for example idvd_vg, the setup window tabs are displayed.

2 Select the **Instrument Options** tab, and a window is displayed showing the instrument states for all instruments used in that measurement. In the DC measurement setups, the instrument states for the DC source/monitor are displayed. In the S-parameter measurement setups, the instrument states for both the DC source/monitor and the network analyzer are displayed.

The instrument state settings are explained in this chapter with their corresponding setups.

Measuring the Characteristic Curves (idvd_vg)

This procedure measures drain current with respect to drain voltage, at several values of gate voltage. The drain voltage is swept from 0V to the upper limit of its normal operating range.

FET Configuration: The input values you set in the following procedure configure the FET as shown in the following figure.
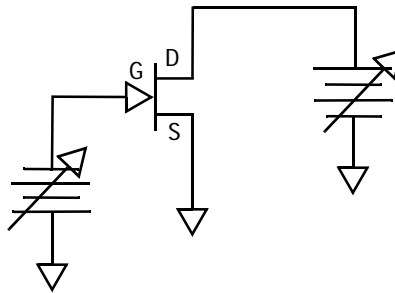
Figure 16   FET Bias Configuration for idvd_vg Measurement

Parameters to Be Extracted

This procedure is done first to determine whether or not the device is good. The data from this measurement will be used to extract beta2 and gamma2, the Curtice Cubic parameters that control the linear and saturation regions of device operation. RDS0 (intrinsic output resistance) is also extracted from this measurement through optimization. (In a Curtice quadratic model, this data would be used to extract the beta, gamma, and alpha coefficients.)

This data can also be used for later comparison with predicted data provided by the extracted model.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Then return to this chapter and continue.

Setting the idvd_vg Input Values

This procedure defines the input signals to be applied to the device under test for this measurement. The actual settings you use will be dependent on the compliance limits for both the device and the SMUs/bias networks, and on the range of measurements you want to take.

If you are measuring a series of devices of the same type and have already set the input values, go directly to the section "Measuring and Plotting" on page 80. Otherwise, select the **idvd_vg** setup (from the DC DUT) and use the following instructions as a starting point to set up the measurement for your particular device. Refer to your device data sheet as needed for specifications and compliances.

Table 5 For the vd input

Set	To
Mode	V (voltage)
Compliance	no greater than the maximum breakdown current value in the device data sheet. With a voltage input, compliance refers to current. Compliance values limit SMU output voltage or current and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used.
Sweep Type	LIN to provide a linear voltage sweep from start to stop voltage values
Start	0.0
Stop	Referring to the data sheet for your device, set the Stop value ≤ the upper limit of the device's normal operating range.
# of Points	Set the # of Points to 21 or less.

The Step Size is calculated automatically in software from the start/stop values and number of points. (Alternatively you can set the Step Size and the # of Points is calculated automatically.) Note, however, that the step size must be an integer multiple of the minimum step size of the SMU (refer to the Agilent 4142 Operation Manual for the SMU specifications).

Table 6   For the vg input

Set	To
Mode	V (voltage)
Compliance	no greater than the maximum allowable gate current value in the data sheet. With a voltage input, compliance refers to current. Compliance values limit SMU output voltage or current and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used.
Sweep Type	LIN to provide a linear voltage sweep from start to stop voltage values
Start, Stop, and # of Points	Set such that three to five values (points) will be measured, in a range from approximately Vgs=Vpinchoff to Vgs=0V. Set Start to a value close to Vpinchoff, but not so close that the curve will be difficult to discern. (Since the data sheet is likely to provide a range of values, these settings may need to be changed after the first measurement.)

CAUTION   In setting vg, be careful not to set gate voltage values high enough to damage the device.

- The Outputs are factory set, and should not be changed, with the exception of Type. This is the type of data to be plotted, M for measured, S for simulated, or B for both. There is a tradeoff in making this choice: if the measurement will be a slow one because of long integration time or large number of points, you can save time by setting Type to M. If the measurement will be fast, you can save the need to change the plot Type later during extraction by setting B. Set it to M for now, to save time in plotting the measured values.

- The Unit names must conform to the names set in the hardware window. Refer to the appropriate chapter as follows:

  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

• "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

They must also correspond with the actual plug-in SMU connections from the Agilent 4142. Set the vd Unit to VD and the vg Unit to VG. Since the source is grounded, only the drain and gate inputs need to be set.

Measuring and Plotting

This procedure triggers the device measurement, controlling the DC source/monitor to supply the signals you defined in the instrument options and the inputs, and monitors the results.

NOTE   Remove any high-intensity light sources such as microscope light before taking a measurement. One simple method of blocking light is to place an opaque plastic box on the fixture directly above the device.

1 From the idvd_vg setup select the **Plots** tab.

2 Select the **id_vs_vd** plot, and **Display Plot**.

3 If you have not already connected the DUT, do so now. *Observe antistatic precautions*.

4 Select **Measure/Simulate > Measure**. *Do not touch* the fixture or the bias networks while the system is measuring. When the measurement is complete, the yellow MEASURING light on the DC source/monitor goes out, the IC-CAP wristwatch icon goes away, and the measured data is displayed in the plot window.

5 The plotted data should resemble the following figure, with the lowest curve close to the X-axis to show the pinchoff characteristics. If the data indicates the device is not operating in its normal range, you may need to change the gate voltage values you set in the inputs. (If more than one curve is low, change vg Start to a higher value. Or if all the curves are high, change vg Start to a lower value.) Then select Measure again.

Plot CGaashf/dc/idvd_vg/id_vs_vd (On)
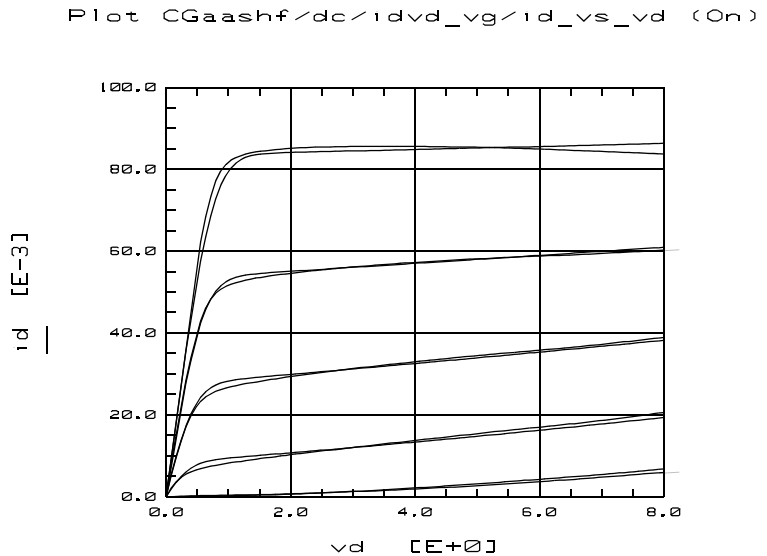
Figure 17   Example idvd_vg Measurement of FET Characteristic Curves

6 If the measured data looks wrong for your device, check the probe contacts, the bias connections, and the measuring instruments. Make sure the device is oriented properly, with the right ports connected to the right terminals. If none of these resolves the problem, the device may be defective. For example, if all the curves are superimposed, the gate may be shorted.

7 If you wish, you can save the measured data for later comparison with data predicted by the extracted model. To do so, from the Main window, select **File > Save As**. Use the File Type suffix .set. Enter an appropriate filename (for example *dc_test*), and press Return or click **OK** to save the file.

8 Close the plot.

Measuring Drain Current vs Gate Voltage (idvg_vd)

This procedure measures drain current as a function of swept gate voltage, at a single drain voltage bias point in the typical operating range of the device. Values of gate voltage are selected to take the device from pinchoff to IDSS.

FET Configuration: The input values you set in the following procedure configure the FET as shown in the following figure.
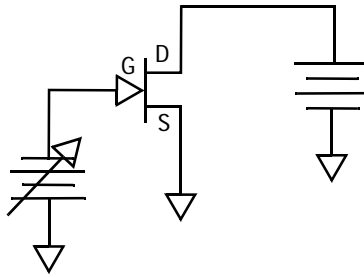
Figure 18   FET Bias Configuration for idvg_vd Measurement

Parameters to Be Extracted

This is a very important measurement, because the A coefficients will be determined from it. The A coefficients form the core of a Curtice cubic model, and therefore must be carefully extracted. Poor measurement and extraction can result in a device model in which the DC performance appears reasonable while the AC performance, specifically S21, is suspect.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

• "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings for the idvg_vd setup as you did for the idvd_vg, *except* set Integ Time to L (long) for maximum averaging, because this measurement uses a derivative process. *Note:* The instrument states need to be set independently for each setup.

Then return to this chapter and continue.

Setting the idvg_vd Input Values

Select the idvg_vd setup and use these guidelines for setting the vd and vg inputs.

Table 7   For the vd input

Set	To
Sweep Type	CON (constant) to provide a constant voltage at one bias value
Value	at or near the typical operating region of the device.

Table 8   For the vg input

Set	To
Sweep Type	LIN (linear)
Start and Stop	Set such that the gate voltage takes the device from pinchoff to IDSS. It is best to set a start value below 0.0V to avoid contaminating lesser values of drain current with the heat generated at IDSS. The Curtice model does not compensate for heating effects, which would cause a downslope in the measured data at the higher values of drain current.
# of Points	Set fairly large (>100) to ensure an adequate set of measurements across the voltage range.

Measuring and Plotting

1 From the Plots tab, select the **id_vs_vg** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The plot will be updated with the measured data, which should resemble the following illustration. Typically Id increases parabolically with the increase in Vg. If not, there may be a problem with the device.
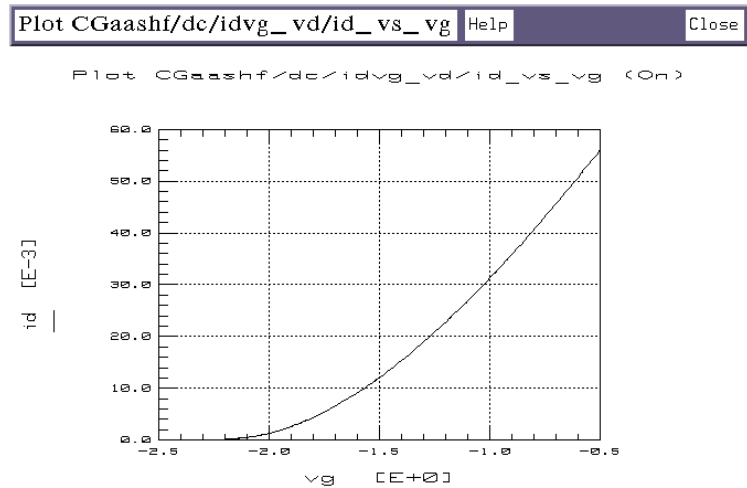
Figure 19 Example idvg_vd Measured Data

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

4 Close the plot.

Measuring Ideality

This setup forces the FET into a forward-biased diode configuration (gate-source junction), by applying a swept voltage at the gate and opening the drain. Gate-source current is measured as the gate voltage is swept.

FET Configuration: The input values you set in the following ideality procedure configure the FET as shown in the following figure.
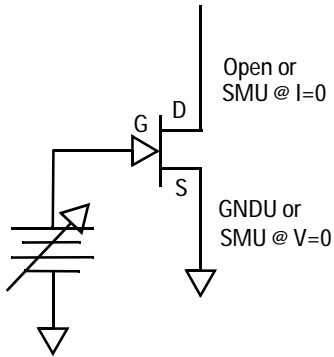
Figure 20   FET Bias Configuration for the Ideality Measurement

Parameters

The data from this DC measurement will be used to extract the diode parameters VBI (built-in voltage), IS (junction saturation current), and N (ideality factor).

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings for ideality that you did for the idvg_vd setup, and set Integ Time to L (long).

Then return to this chapter and continue.

Setting the ideality Input Values

CAUTION	Since the gate-source junction is forward-biased in this configuration, be particularly careful in setting compliance values.

Select the ideality setup and use these guidelines for setting the vg and id inputs.

Table 9   For the id input

Set	To
Mode	I
Value	0.0 to float the drain
Sweep Type	CON (constant)
Compliance	In current mode, compliance refers to voltage. If the device data sheet specifies a maximum drain voltage, use that value for Compliance.

Table 10   For the vg input

Set	To
Compliance	a good rule of thumb is to use 0.1 mA gate current per micron of gate width (for example 50 mA for a 500 um device). Allowing too high a current compliance can overstress the device and degrade its performance, even though the results may not be obvious.
Sweep Type	LIN (linear) for a swept voltage from Start to Stop values

Table 10   For the vg input

Set	To
Start and Stop	Start with conservative values for gate voltage to prevent possible damage to the device. They can quickly be changed after the first measurement if necessary. The FET data sheet is unlikely to specify diode characteristics, so start with best-guess values. Begin by setting a vg Stop value of 500.0m, and increase it if appropriate until the measured data shows the characteristics explained under "Measuring and Plotting" on page 87 (The increase must be an integer multiple of the SMU step size.) A typical gate voltage range is from 100 mV to 800 mV.
# of points	Use a large enough # of points to ensure distribution of measurements across the exponential region of the diode characteristics.

Measuring and Plotting

1 From the Plots tab, select the ig_vs_vg plot, and **Display Plot**.
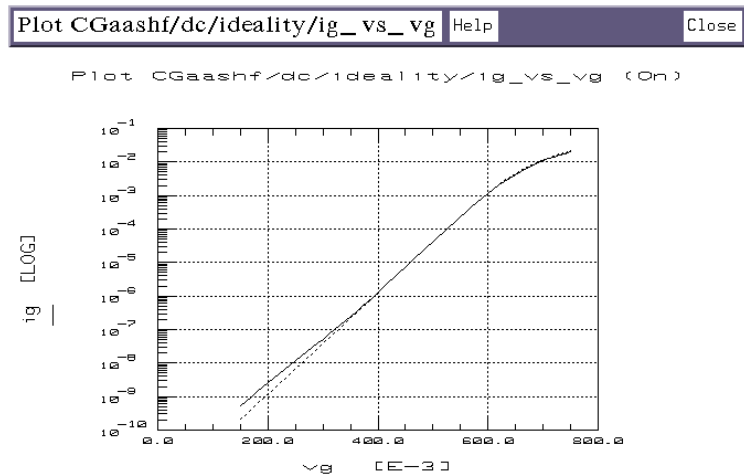
Figure 21   Example Measured ideality Data

2 Select **Measure/Simulate > Measure**. The plotted data should be similar to the illustration. (Note that Ig is plotted on a log scale to show the exponential behavior of the diode junction.) The current should be very low at the low end, and the curve should bend at the high end where the parasitic gate and drain resistances are having an effect. The trace should be smooth and noise-free if a long integration time was used. Noise at the low end occurs if a shorter integration time is used.

3 The bend at the high end of the curve defines where the parasitics start dominating, and sets the bounds for extraction. If the displayed data does not show this bend, go back to the inputs and increase the Vg Stop value slightly (perhaps by 100 mV). Repeat the measurement. Iterate as necessary, gradually increasing Vg Stop until the bend in the trace does appear.

4 The other plot is used later, in the extraction process.

5 Close the plot.

Measuring Breakdown Characteristics (Optional)

Breakdown is not normally a factor in small-signal device operation, typically only occurring if the device is driven hard. The usefulness of this measurement is limited, since the model's extraction of VBR (breakdown voltage) defines a hard breakdown point, while MESFETs typically have a soft breakdown characteristic. However, breakdown is one source of nonlinearity, so you may want to measure and model it. Be aware, however, that it may cause prediction of false harmonic values when the model is used in a simulation.

In this measurement, the source is open and the drain grounded. The gate current is monitored as gate voltage is swept from a positive voltage in the saturation region to a negative voltage beyond the expected reverse breakdown point. If you have a standard system with two SMUs, either disconnect the GNDU SMU from the source and leave it open, or reverse the source and drain connections (GNDU at the drain and SMU2 at the source).
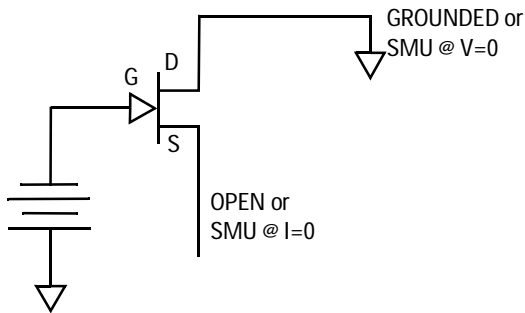
Figure 22   FET Bias Configuration for breakdown Measurement

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings for breakdown as you did for ideality. Integ Time can be set to M (medium).

Then return to this chapter and continue.

Setting the breakdown Input Values

Select the breakdown setup and use these guidelines for setting the vg and vd inputs.

Table 11   For the vg input

Set	To
Compliance	Set a small value to prevent damage to the device in the reverse breakdown region. 500A is appropriate

Table 11   For the vg input

Set	To
Sweep Type	LIN for a linear voltage sweep from Start to Stop
Start	a positive value in the saturation region
Stop	a negative voltage beyond the expected reverse breakdown point

Table 12   For the vd input

Set	To
Compliance	small
Sweep Type	If you disconnected the source and left it open, set Sweep Type to CON (constant), with a value of 0.0. (Alternatively, connect the GNDU SMU to the drain.)

Measuring and Plotting

1 From the Plots tab, select the ig_vg plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The measured data should be similar to that shown in the following figure. VBR (breakdown voltage) is the gate voltage Vg value at which gate current Ig stops increasing.

Plot CGaashf/dc/breakdown/ig_vg   Help   Close

Plot CGaashf/dc/breakdown/ig_vg (On)
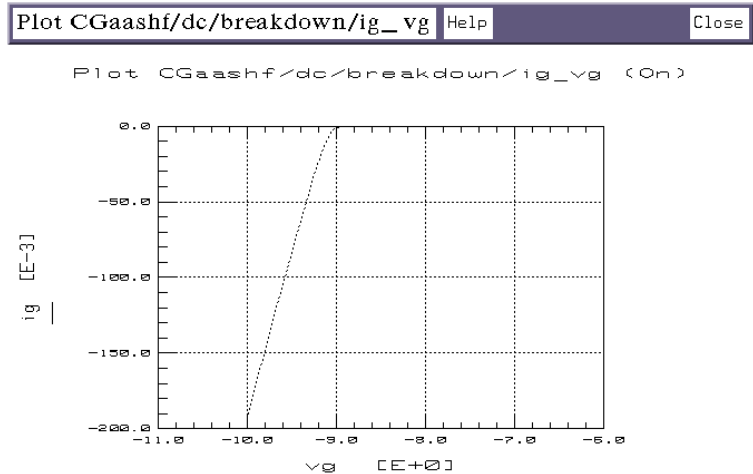
Figure 23   Example breakdown Measurement Data

3 Close the plot.

Calibrating the Network Analyzer

It is important to calibrate the network analyzer before you perform the S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and extraction.

For this procedure, the network analyzer must be calibrated over a broadband frequency range at least equal to the operating range of the device. No subset calibration is required.

For step-by-step calibration procedures, refer to the appropriate chapter as follows:

• "Calibrating the Network Analyzer" on page 589 (if you have an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

• "Calibrating the Network Analyzer" on page 631 (if you have an Agilent 8753-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Then return to this chapter and continue.

NOTE   IC-CAP requires values to be entered into the software regarding calibrations, instrument states, and measurement values. It is important that values entered in software correspond with actual instrument settings.

Making the S-Parameter Measurements

The swept S-parameter measurements in the next two setups provide the data that allows extraction of parameters related specifically to the high-frequency performance of the device. A swept-frequency signal is applied to the gate, and the device S-parameters are measured at the drain.

Setting up the measurement conditions involves processes similar to those for the DC measurements: defining the instrument states and setting the input values for each measurement.

Measuring S-Parameters vs Frequency

This procedure measures S-parameters across a calibrated microwave frequency range, with a fixed bias value in the typical linear operating region of the device.

FET Configuration: The input values you set in the following s_vs_f procedure configure the FET as shown in the following figure.
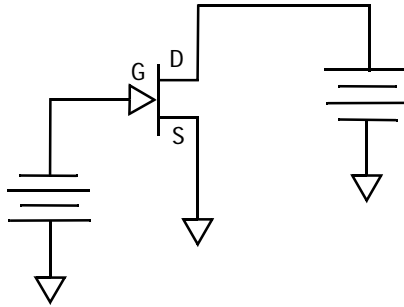
Figure 24   FET Bias Configuration for the s_vs_f Measurement

Parameters

The data from this measurement will be used to extract RIN, RDG, CDGN, RDGN, CDG, CGS0, A5, and RC.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Instrument States for Swept Frequency Measurements" on page 624 (Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Set the Cal Set No to the cal set or register where you stored the broadband calibration you performed earlier. The procedures in this manual use cal set #1.

Then return to this chapter and continue.

Setting the s_vs_f Input Values

Connect the network analyzer RF input cables to the RF IN ports of the bias networks and select the **s_vs_f** setup.

Table 13   For the freq input

Set	To
Mode	F
Sweep Type	LIN (linear) for a linear measurement sweep across the frequency range.
Sweep Order	1
Start, Stop	Set the Start and Stop values to correspond with the frequency you defined in your broadband calibration.
# of Points	The same as in the calibration.

Table 14   For the vd input

Set	To
Mode	F
Sweep Type	CON (constant), to provide fixed bias values of both gate and drain voltage
Compliance	no greater than the maximum breakdown current value in the device data sheet
Value	bias the device in a typical linear operating region of the device

Table 15   For the vg input

Set	To
Compliance	no greater than the maximum allowable gate current value
Sweep Type	CON (constant), to provide fixed bias values of both gate and drain voltage
Value	bias the device in a typical linear operating region of the device

- The Outputs are factory set, and should not be changed, except for Type. To reduce time in plotting all four S-parameters, you can set Type to M (medium) for now.

Measuring and Plotting

1 Select **Plots > Display All**. This displays all the plots at the same time. Alternatively, you can display them one-by-one using the method explained in the other measurement procedures.

2 Select **Measure/Simulate > Measure**. The plotted data should be typical for your device. The illustrations below show measured S-parameter data for an example device. If the results are unexpected, recheck the values you set in the inputs and instrument options. In particular, be sure the instrument states and input stimulus signals match the values set in the network analyzer calibration.

3 If you change the inputs or instrument states, repeat the measurement. If the data still looks abnormal, the device may be defective.

4 Save the measured data using **File > Save As** and a filename with the suffix .*set*.

5 Close the plots.

Measuring Parasitic Resistances and Inductances

This procedure uses S-parameter measurements to determine the device parasitic resistance and inductance values. It measures S-parameters vs frequency, with both the gate-source and gate-drain junctions strongly forward biased, and fixed values of gate and drain current. Drain current is set to one half gate current, to force Is = Id: this makes the intrinsic Vds zero, removing the channel resistance. The device can thus be modeled as a T-network.

FET Configuration: The input values you set in the following parasitics procedure configure the FET as shown in the following figure.
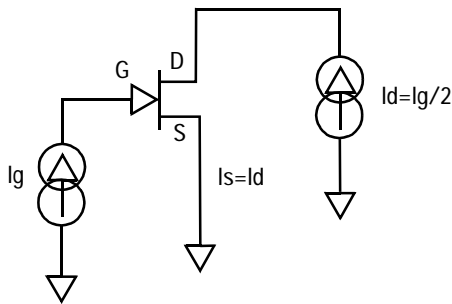
Figure 25   FET Bias Configuration for parasitics Measurement

This measurement is done last because of the high gate current applied to the device. However, this high current should not be a problem because the power dissipation is not correspondingly high with Vds forced to zero. Theoretically, if the parasitic resistance were large enough for heat to be generated, the device could be damaged, but this situation is highly unlikely.

Parameters

The data from this measurement will be used to extract all the parasitic resistance and inductance values: RG, RD, RS, LG, LD, and LS.

Defining the Instrument States

Follow the same procedure you used in the s_vs_f measurement. Use the same settings.

Setting the parasitics Input Values

1 Select the **parasitics** setup.

2 Set the input values under freq the same as for the s_vs_f measurement, as you will be using the same network analyzer calibration.

3 Set the id Compliance as high as the device maximum Vds value or higher. The voltage compliance should not be so low as to inhibit current flow in this measurement.

4   Set the ig Compliance as high as the device maximum Vgs value or higher.

5   Set the Sweep Type to CONstant for both id and ig to provide fixed current values.

6   Set the id current Value to ID and the ig Value to IG. Numerical values are assigned to these variables as follows:

   •   To ensure that the Is = Id = Ig/2 relationship is maintained, the ratio between gate current and drain current is defined in the setup variable table. This allows the gate current to be set to a numerical value and the drain current to be specified with respect to the gate current.

   •   Select the **Setup Variables** tab. A variable table is displayed where you can define Ig and Id.

   •   Set the IG current value based on gate width: 0.1 to 0.2 mA per micron. Thus for a device with 500 um gate width, Ig would be 100 mA. This may seem like a lot of gate current, but the larger Ig is, the smaller the dynamic impedance becomes.

   •   Set the ID current to –IG/2, to force Is = Id.

Measuring and Plotting

1   From the Plots tab, select the **R_data** plot, and **Display Plot**.

2   Select **Measure/Simulate > Measure**.

3   To make sure the SMUs did not compliance-limit during this measurement, check the UNIX window for an error message to that effect. If the SMUs did reach compliance limit, the current may not have been balanced in the drain and source, which will cause misleading extraction results. In this case, go back to the inputs and increase the ig and id Compliance values by as much as 10 to 20%.

4   The measured data should resemble the following illustration. The annotation shows the resistance data as the real part of the device impedance Z. The displayed traces are the measured values for Rs, Rd, and Rg.
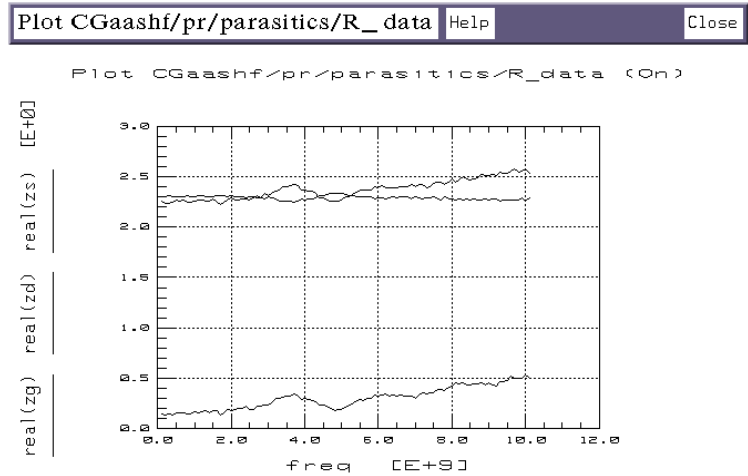
Figure 26   Example Parasitic Resistances

5 Select the **L_data** plot, and **Display Plot**. The plotted data should resemble the illustration. The annotation shows the inductance data as the imaginary part of the device impedance Z. The displayed traces are the measured values for Ld, Lg, and Ls.
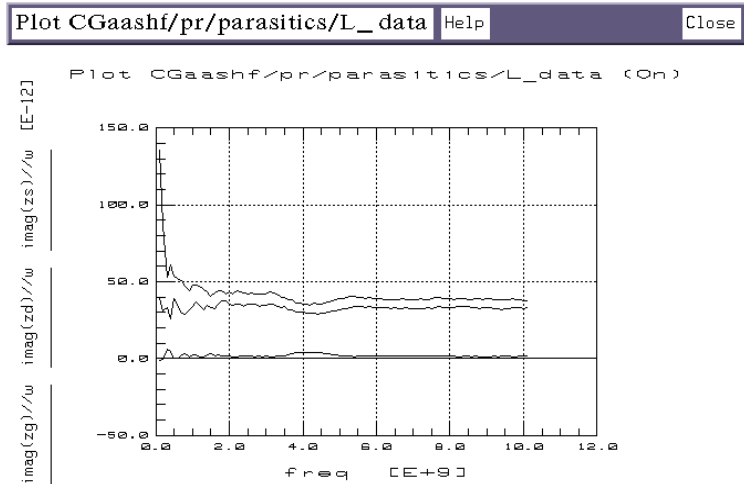
Figure 27   Example Parasitic Inductances

6 Leave the parasitics setup open, as it is the first setup used in the extraction process.

Parameter Extraction

Now that all the measurements have been made, the extraction process can begin. There is no single clear and exact procedure for extracting parameters: the process is very iterative and subjective. The goal is to create a list of device parameters that can be used in a model to accurately predict performance. A key to success is that the parameters must be realistic. Because of the inherent coupling of the model equations, a variety of parameter value combinations will result in a good match between simulated and measured data.

The specific measurement setups are defined for the purpose of decoupling the model equations as much as possible. In this way, the effects of individual parameters or small groups of related parameters can be isolated, and their values determined with reasonable certainty. However, no matter how good the measurement setup, there is always some interaction between

parameters. Therefore, during the extraction process, the validity of the extracted parameters must be constantly scrutinized.

The ability to discern reasonable values from questionable ones comes from knowledge of device physics, knowledge of the particular device, and individual experience. Parameter extraction is still very much an art.

Parasitic Extraction

In all the equations governing the behavior of the model, the voltages are assumed to be intrinsic. There is no direct way to measure or set the intrinsic voltages, so they must be determined indirectly. To accomplish this, the parasitic resistances and inductances must first be very accurately determined. Once these are known, the intrinsic voltages can be calculated from these parasitic values and the known currents, using the following equations:

DC:

VGSint = VGSext - Ig · RG - Is · RS

VDSint = VDSext - Id · RD - Is · RS

AC:

vgsint = vgsext - ig(RG + jωLG) - is(RS + jωLS)

vdsint = vdsext - id(RD + jωLD) - is(RS + jωLS)

These intrinsic voltages are used by the Curtice cubic model to predict device performance, so they must be known before any other parameters can be extracted.

Selecting the Simulator

One last step remains before the actual extractions: to check that the selected simulator is compatible with the model. The only simulator that supports the Curtice GaAs model is hpspice.

Select the **Model Variables** tab. A variable table is displayed, showing the name of the currently selected simulator. If the SIMULATOR name is not hpspice, then type `hpspice`.

Displaying the Parameters Table

The CGaashf Model Parameters table lists all the parameters that will be extracted, and initially displays default values. As you perform the individual extractions, the default values change to extracted values. Throughout the extraction process, you will use the Parameters table to read extracted values, and to enter or modify values to adjust interactive parameters for best agreement of simulated with measured data. In addition, you can set limits by entering minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values. To view the parameters, click the **Model Parameters** tab.

Extracting Parasitic Resistances and Inductances

This procedure extracts the parasitic gate, drain, and source resistances (RG, RD, RS) and gate, drain, and source inductances (LG, LD, LS). These parameters are extracted from the S-parameter data measured with the parasitics setup.

1 Locate the default values for RG, RD, RS, LG, LD, and LS in the Parameters table.

2 Select the **parasitics** setup from the DUT/Setup panel.

3 From the Plots tab, select **R_data > Display Plot**, and **L_data > Display Plot**.

4 From the Extract/Optimize tab, select the **autoextract** transform and **Execute**. The transform calculates the parasitic values with frequency and enters them in the Parameters table. In the plots, the resistance data displayed is the real part of the device impedance Z, and the inductance data is the imaginary part of the impedance.

5 Select the **zg** transform and **Execute**. The transform data is added to the R_data and L_data plots.

6 To verify the RG and LG data values in the Parameters table against the values plotted, position the pointer at a frequency of interest or on some approximately median point on the trace, and click middle. A readout of the trace value (Y) at that point is listed above the plot: compare this value with the value in the Parameters table.

7 If the trace value you selected and the Parameters table value
do not closely correspond, it may be because a portion of the
trace is noisy. In particular, the inductance values at the
lower frequencies are likely to be noisy and less valid. You
may wish to change the Parameters table value to more
closely represent a linear portion of the trace. The parasitic
values calculated by autoextract are averaged across the
entire frequency range of the measurement by default. To
average across only a portion of the trace you can use the
X-low/X-high function.

• Select the **Setup Variables** tab. A variable table will appear,
listing X_LOW and X_HIGH with their default or current
frequency values.

• From the plotted data choose a linear portion of the trace.
Put the mouse pointer at the low end of that portion and
click left, then put the pointer at the high end and click left
to make a box on the trace, as shown in the illustration.
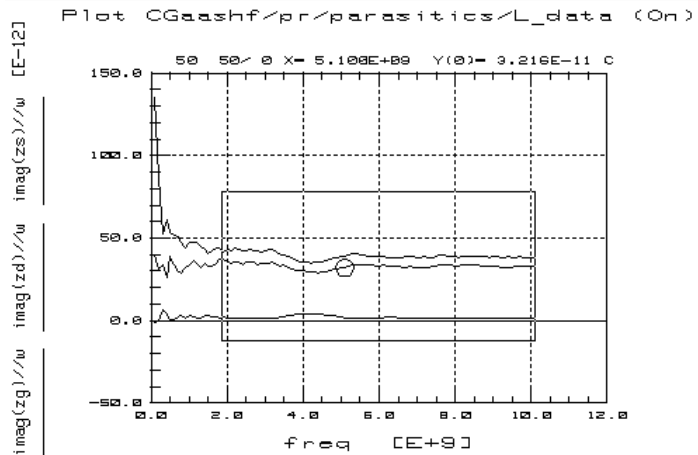
Figure 28   Selecting a Linear Portion of the L_data Trace

• From the plot menu select **Options > Copy to Variables**. This
changes the X_LOW and X_HIGH values in the table to the
values you selected on the trace.

- Repeat **autoextract > Execute**. The transform recalculates the parasitic values using the new X-low/X-high values.

8 Select **Extract/Optimize > zd > Execute**. To verify the drain resistance and inductance values (RD and LD) in the Parameters table, follow the same steps as for the gate parasitics.

9 Select **zs > Execute**. To verify the source resistance and inductance values (RS and LS) in the Parameters table, follow the same steps as for the gate parasitics.

10 Close the plots.

Extracting the Gate Diode Parameters

This procedure extracts the parameters N (ideality factor), VBI (built-in voltage), and IS (junction saturation current). These parameters are extracted from the DC data measured with the ideality setup.

N (Ideality Factor)

1 Scroll the Parameters table until it shows the value for ideality, represented as NMF1.N.

NOTE   Several of the parameter names in the table are preceded by NMF1.

2 Select **DUTs/Setups > ideality**.

3 Select **Plots > n_vs_vg > Display Plot**.

4 Select **Extract/Optimize > extract > Execute**.

5 Select the **n** transform, and **Execute**. This calculates the ideality factor and enters it into the Parameters table.

6 The data displayed in the n_vs_vg plot should resemble Figure 29. N is computed as the minimum value along the linear region of the logarithmic curve (shown in the box in Figure 29). Beyond the linear region, the parasitic values begin to dominate, and N is no longer identifiable.

7 To check the N value in the Parameters table against a value on the trace, put the mouse pointer on the linear portion of the trace and click middle for a readout of the Y-axis value at that point.

8 If the n_vs_vg plot looks significantly different from the following figure, the N value in the Parameters table may not be valid. For example, if noise is present on the trace below the linear region of the curve, N may have been computed as the minimum value of the noise. In this case, use the X-low/X-high function to get a more valid value.
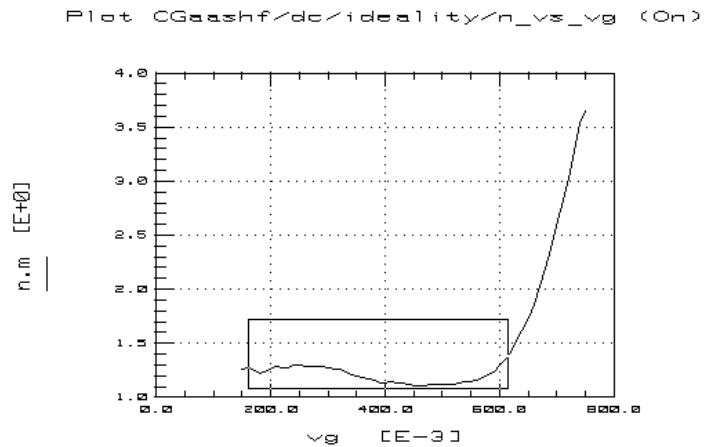
Figure 29   Linear Portion of the n_vs_vg Plot

9 To compute N over only the linear portion of the trace using X-high/X-low:

- Select **Setup Variables** to open the X-high/X-low variable table.

- In the Value field (right side of the table) for OVERRIDE_LIMITS, change FALSE to TRUE, to set override limits to true.

- Make a box on the trace by clicking left on the ends of the linear portion.

- From the plot menu select **Options > Copy** to Variables to change the X-low/X-high values to the trace limits you selected.

- Select **Extract/Optimize > extract > Execute** to recalculate the N value using the new X-low/X-high values.

- Check the value of NMF1.N in the Parameters table.

10 If no linear region is shown on the trace, a valid ideality factor cannot be computed from it. In this case, you will need to manually enter a value of 1 or 0. Since the contribution of N to the gate current is relatively small, it is best when the data is very nonlinear to eliminate the contribution of the N factor by using a value of 0. To change the NMF1.N value in the Parameters table, move the mouse pointer over the current entry and click left to highlight it, then type in the new value and press Return.

IS (Saturation Current)

1 Select **Plots > ig_vs_vg > Display Plot**. The data displayed should resemble the following figure, with two plotted functions of gate current Ig vs gate voltage Vg. The solid line is the measured data, and the dashed line is the simulation.
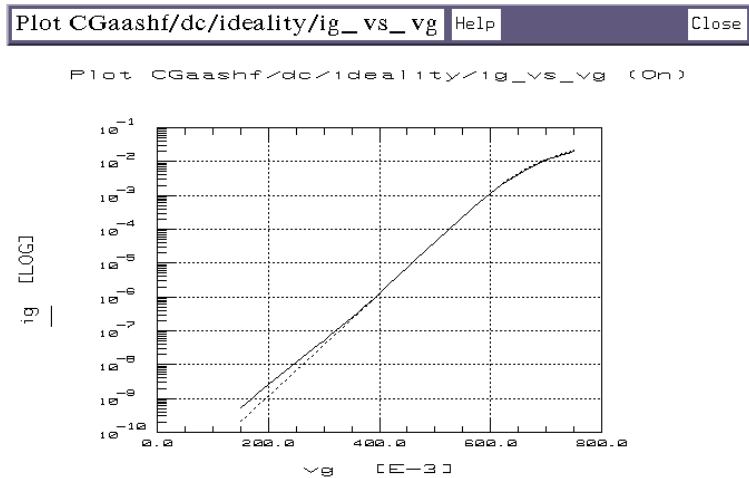
Figure 30   Measured and Simulated Saturation Current (IS) Data

2   Scroll the Parameters table until it shows the value for NMF1.RF.

3   Select **DUTs/Setups > Extract/Optimize > rf_off > Execute**. The value of NMF1.RF in the Parameters table will change to 0.

4   Scroll the Parameters table until it shows the value for NMF1.IS.

5   Select **Extract/Optimize > opt_is > Execute**. Observe the plot and see the simulated data converge on the measured data as the value for IS is optimized. (Note that the opt_is transform optimizes the linear data for which the ideality factor is valid.)

6   Observe the change in the Parameters table value for IS.

7   The parameter value for VBI is also changed.

8   Close the plots.

Extracting the A Parameters and VDS0

This procedure extracts and optimizes the A0, A1, A2, and A3 coefficients that describe the threshold characteristics of the device. VDS0 is the value of VDS at which A0 through A3 are determined; its default value is the value of Vd specified in the idvg_vd inputs.

These parameters are extracted from the DC data measured with the idvg_vd setup.

1   Scroll the Parameters table until it shows the default values for A0, A1, A2, and A3.

2   Select **DUTs/Setups > idvg_vd**.

3   Select **Plots > id_vs_vg > Display Plot**.

Plot CGaashf/dc/idvg_vd/id_vs_vg (On)
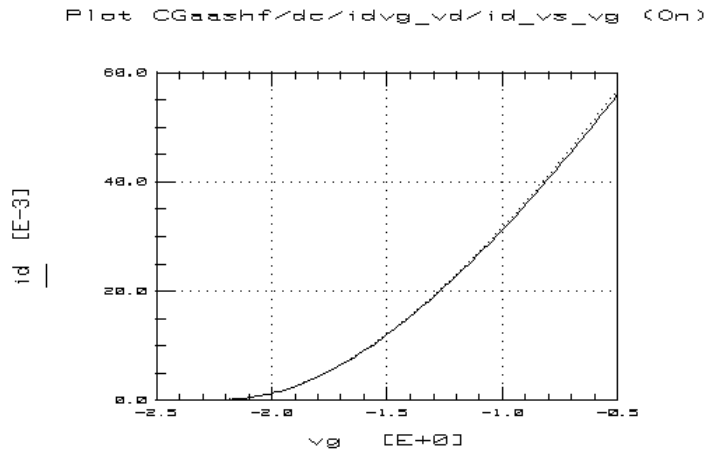
Figure 31   A Parameter Data After Optimization

4 Select **Extract/Optimize > extract > Execute**.

5 The transform calculates the values of A0, A1, A2, A3, and VDS0 and enters them into the Parameters table.

6 Scroll the Parameters table until you find the value for NMF1.VDS0. Manually enter this same value into the Parameters table for VDSDC (you will need to scroll the Parameters table again to find it).

7 Select **Measure/Simulate > Simulate**. The plot should now display a solid-line measured curve and a dashed-line simulated curve.

8 Select **Extract/Optimize > opt_cubic > Execute**. Watch the simulated data converge on the measured data as the values for the A coefficients are optimized (as in Figure 31).

9 Select **Model Parameters**, and observe the new Parameters table values for the A coefficients and VDS0.

10 Close the plot.

Extracting Beta and Gamma Parameters and RDS0

This procedure extracts beta2 and gamma2, the Curtice cubic parameters that control the linear and saturation regions of device operation. Gamma2 dominates in the linear region and determines the transition from linear to saturation; beta2 dominates in the saturation region.

Also extracted through optimization is RDS0, the intrinsic output resistance. RDS0 dominates at both the low and high values of Vds relative to VDS0 (that is, Vds ≤ VDS0 and VDS0 ≤ Vds).

These parameters are extracted from the DC data measured with the idvd_vg setup.

1 Scroll the Parameters table until you find RDS0. Manually set an RDS0 value of 1K ohms, as the initial value to be used in the optimization.

2 Select the **idvd_vg** setup.

3 Select **Plots > id_vs_vd > Display Plot**.

4 Select **Extract/Optimize > extract > Execute**.

5 Select **Measure/Simulate > Simulate**.

6 The plot should now display a family of solid-line measured curves and dashed-line simulated curves, as in the following figure.
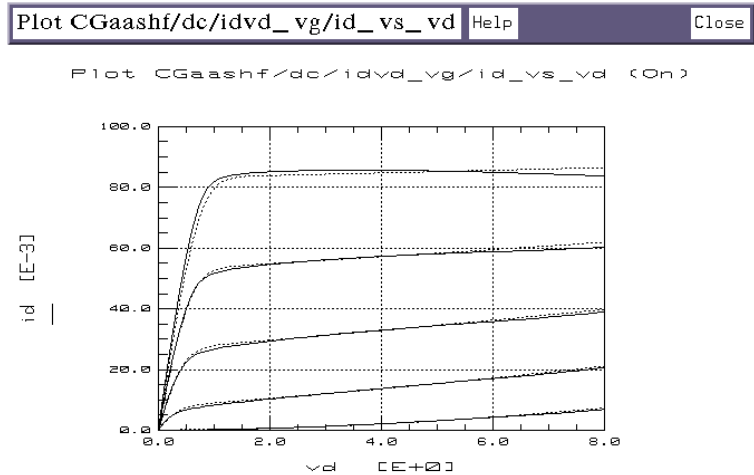
Figure 32   Measured and Simulated Data for Beta, Gamma, and RDS0
  Extraction

7 Select **Extract/Optimize > opt_cubic > Execute**. Watch the
    simulated curves converge on the measured curves as the
    beta and gamma coefficients are optimized.

8 Scroll the Parameters table and observe the new values for
    BETA, GAMMA, and RDS0.

9 Close the plot.

Extracting the AC and Capacitance Parameters

This extracts the parameters RIN, RGDN, CGDN, CDS, CGS0,
A5, and RC. These parameters are extracted from the
S-parameter data measured with the s_vs_f setup. The
transforms calculate the intrinsic admittances vs frequency.

In an extraction of admittances vs frequency, the linear region
of the data may be bounded by nonlinearities at the ends of the
frequency range. Capacitance tends to dominate at lower
frequencies, and inductance at higher frequencies. The most
appropriate values for the extracted parameters occur in the

linear resistive region of the data. The X-low/X-high function lets you define the boundaries of the linear region and perform the transform over only that region.

1 Select the **s_vs_f** setup.

2 Select the **Plots** tab, then select and display the following plots: Rs, Cs, tau_a5, and CGS0.

3 Select the **Extract/Optimize** tab, then select and execute the following transforms: **ygs**, **ygd**, **yds**, **a5**, **cgs0**, and **ygm**.

4 Examine the plots and use the X-low/X-high function if necessary to limit the range of the extractions that will follow:

- Select **Setup Variables** to open the X-high/X-low variable table.

- Click left at the ends of the linear region to make a box on the trace.

- From the plot menu select **Options > Copy to Variables** to change the X-low/X-high values to the trace limits you selected.

- Select **Extract/Optimize > extract > Execute** to recalculate the parameter values using the new X-low/X-high values.

5 Select **extract_RC > Execute**. This transform optimizes RC to minimize the difference between the measured real yds and the simulated real yds.

6 Verify the values of the extracted parameters listed in the Parameters table against the measured data in the plots.

- The Cs plot shows the value for CGDN as imag(ygd), the imaginary part of the measured gate admittance; and CDS as imag(yds), the imaginary part of the measured drain admittance.

- The Rs plot shows RIN as real(ygs), the real part of the measured gate admittance; and RGDN as real(ygd), the real part of the measured drain admittance.

- The CGS0 plot shows CGS0 as real(cgs0).

- The tau_a5 plot shows A5 as imag(a5).

Verifying the AC and Capacitance Extractions

1 Select **Plots > S11 > Display Plot**. Repeat this for **S12**, **S21**, and **S22**.

2 The plots show the measured and simulated S-parameter values. If the extraction is good, the simulated traces should converge toward the measured traces, as in the S-parameter plots below.

3 If the simulated traces do not match the measured traces well, recheck the extracted R, C, and L values.

4 Close the plots.

This ends the extraction process.

Storing the Model

This procedure stores the model in IC-CAP, including all the device-specific measured and simulated data. Once you have developed the device model, it can be used as a component in the Agilent RF and Microwave Design System (MDS). You can use it for circuit simulation, inserting it into circuits under design to emulate the characteristics of your device.

If you will be modeling large numbers of devices, it is worthwhile to organize your directory structure according to device type, geometry, and batch or wafer.

1 Use the UNIX directory you created at the beginning of the procedure, when you first opened the *CGaashf.mdl* file.

2 From the CGaashf:1 main menu select **File > Save As**.

3 A dialog box is displayed, as illustrated.
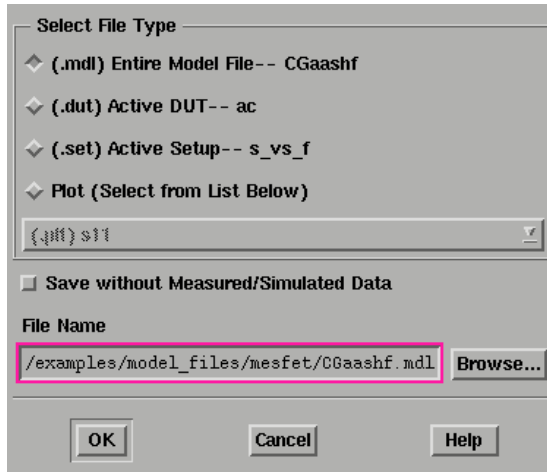
Figure 33 Save As Dialog Box

4 Select the File Type (*.mdl* for a complete model) by clicking the corresponding radio button.

5 Select a new filename that associates this model with the device, for example *fet5.mdl*. Type the full directory pathname and the filename into the dialog box. Select **OK** to store the data and close the dialog box.

Storing the Parameters Table

The Parameters table lists all the parameters extracted for a device model. You can save the Parameters table for a specific device. The illustration shows a portion of the table.
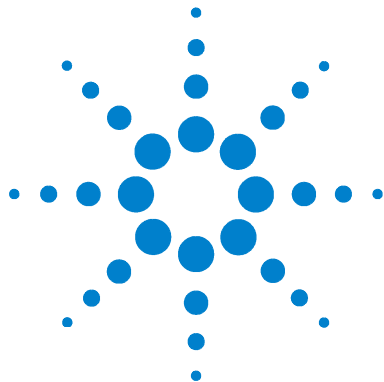
Figure 34   Top Portion of the Model Parameters Table

1 To save the Parameters table select **File > Save As**.

2 In the dialog box, select the File Type *.mps* for model parameter set.

3 Type in a filename or use the one in the File Name field.

4 Select **OK**.

You can now recall the same set of device parameters any time you wish.

3
High-Frequency BJT (Gummel-Poon) Model

The Agilent 85193A high-frequency BJT model is a three-terminal version of the Gummel-Poon model, with a measurement and extraction methodology that has been developed specifically for high-frequency devices.

The high-frequency BJT model provides improved accuracy for modeling the AC parameters of a high-frequency device, by using a network analyzer to measure the device S-parameters. Additionally, because accurate capacitance measurements are extremely difficult at high frequencies, the software computes the model junction capacitances from the measured S-parameter data, rather than using a CV meter.

This chapter provides example procedures to enable you to use the software as a tool to measure your devices and extract their model parameter values. They take you step-by-step through a BJT measurement and parameter extraction. The procedures are arranged in the most logical sequence to allow earlier measured data or extracted parameters to be used as a basis for later measurements or extractions. The sequence also requires the least number of configuration changes.

The procedures use a series of different setups to measure current or voltage versus bias under different bias conditions. The measured values are later used in the extraction process to isolate individual parameters. In the BJT model, the parameters

Agilent Technologies

are very interdependent because of the inherent coupling of the model equations. The purpose of the different measurement setups is to decouple the SPICE equations as much as possible.

Getting Started in IC-CAP

These procedures assume the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System."
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System."
  - Wiltron 360-based system—Refer to your Wiltron documentation

- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.

- The IC-CAP software has been properly installed on the computer.

- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename a medium-power SMU connected to the device base as SMU1

Rename a high-power SMU connected to the device collector as SMU2

Rename the GNDU unit (which must be connected to the emitter) as GND.

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on opening the IC-CAP software and accessing the model file you need. Find the high-frequency BJT model file under:

/examples/model_files/bjt/bjt_ncehf.mdl

Then return to this chapter and continue with the procedures here.

Before you continue, create a new directory (using the mkdir command), to store the data and model files that will be generated, and then change to that directory.

Using the High-Frequency BJT Model

To effectively isolate the individual parameters of a BJT, some device measurements require that the collector is biased and the emitter grounded, while some require that the emitter is biased and the collector grounded. These procedures assume that in-fixture measurements are made with the device mounted in the fixture in a common-emitter configuration as shown in the following figure. In some measurements, the collector is synchronized to the emitter with an offset, to provide data equivalent to that of a common-collector configuration.
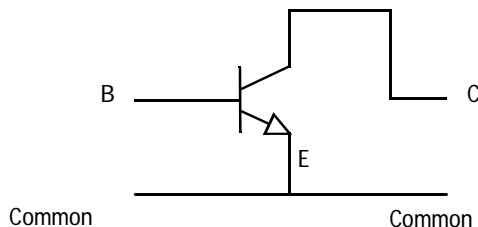
Figure 35   BJT in Common-Emitter Configuration

NOTE

An earlier version of the Agilent 85193A high-frequency BJT model made use of a BJT switching adapter to switch the device between biased-collector and biased-emitter configurations. Now a synchronized sweep (SYNC) is used to synchronize the base and collector SMUs together, allowing all measurements to be made using a common-emitter configuration thus eliminating the need for the switching adapter. The earlier model, bjt_nhf, can still be used, along with the switching adapter, if you prefer.

The procedures in this chapter include detailed instructions for:

• Setting the instrument states for each measurement

• Setting the input values for each measurement

• Making the measurements and qualifying the results

- Extracting DC, capacitance, and AC parameters; optimizing the results
- Verifying the simulated performance against measured data
- Adjusting the extracted parameters for best model accuracy

NOTE   Measurement parameters and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct method or results.

The procedures start with example values for IC-CAP measurement inputs. They provide guidelines on setting the values to measure your device. However, the values you use will be based on your knowledge of your device and on information provided in its device data sheet, as well as on the guidelines provided here: they may or may not correspond with the examples. In some cases, it may be necessary to set the input values, make a measurement, and then reset the inputs based on the measurement results. The procedures give suggestions on judging the applicability of the input values and measured data.

The illustrations of plotted data are provided as visual examples of possible measurement results. They are not intended to represent a sequential measurement of one device, nor do they necessarily correspond with the example measurement inputs described in the procedures. The plots can be used to check the reasonableness of your own measured data, which should appear similar in shape but not necessarily coincident in values.

In addition, the IC-CAP software itself includes measured data for some of the setups. This data is from the measurement of a single device, and shows another example of possible results. Many, though not all, of the input settings for this measurement correspond with the example settings.

The DUTs and Setups

The DUT/Setup panel is the central access point for the measurement and extraction procedures. The DUTs (for example DC) indicated by the inverted triangles, are groupings of similar measurement setups used to extract related model parameters. The setups contain the information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.
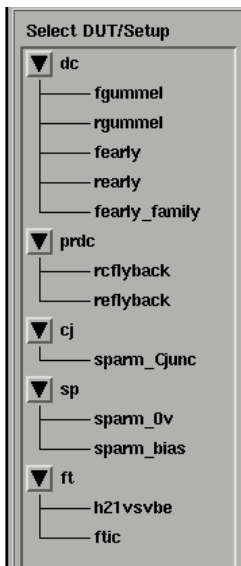
Figure 36   The DUT/Setup Panel

Making the DC Measurements

The DC measurement procedures measure DC current or voltage versus bias under different bias conditions. Setting up the measurement involves two main processes for each measurement setup: defining the instrument state for the DC source/monitor, and setting the inputs for the device measurement.

Leave the network analyzer disconnected from the bias networks while you perform the DC measurements, with the RF IN connectors of the bias networks terminated to prevent possible bias oscillation.

Defining the Instrument States

For each of the measurement setups, it is necessary to define the instrument states (options) for the measurement instruments that will be used in that setup. The instrument states need to be defined individually for each setup. Different setups use different measurement instruments. However, with minor exceptions that will be explained, the instrument states for the same measurement type must be used for all the setups that use that measurement type.

To define the instrument state for any given setup:

1 Select the setup, for example fgummel.

2 Click the **Instrument Options** tab, and the window changes to display the instrument states for all instruments used in that measurement. In the DC measurement setups, the instrument states for the DC source/monitor are displayed. In the S-parameter measurement setups, the instrument states for both the DC source/monitor and the network analyzer are displayed.

The instrument state settings will be explained in this chapter with their corresponding setups.

Making the Forward Gummel Measurement

The forward and reverse gummel measurements plot the operating range of the base and collector currents with the transistor in the forward active and reverse active modes. All coefficients of the DC model equations are derived from the fgummel and the rgummel data.

This procedure measures Ic and Ib with respect to Vbe. The base and collector are swept across a range large enough that both high-level and low-level effects occur. This measurement should always be done first because its result is used to properly define the ranges of subsequent measurement setups.

BJT Configuration: The input values you set in the following fgummel procedure configure the BJT as shown in the following figure.
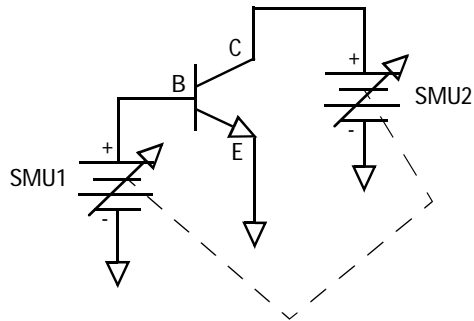
Figure 37   BJT Configuration for fgummel Measurement

Parameters

The data from this measurement will be used to extract IS, BF, NF, ISE, NE, and IKF. It may also need to be used to interactively modify other parameters such as RE, RC, RB, and VAR.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Defining the DC Source/Monitor Instrument State" on page 621 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

However, for the forward gummel measurement, set Integ Time to L (long) to reduce noise at low current values. Then return to this chapter and continue.

Setting the fgummel Input Values

This procedure defines the input signals to be applied to the device under test for this measurement. The actual settings you use will depend on the compliance limits for both the device and the SMUs/bias networks, and on the range of measurements you wish to take.

You provide compliance values to limit SMU output voltage or current and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes, if used. With a voltage input, compliance refers to current. Refer to your device data sheet as needed for specifications and compliances. From the DC DUT select the **fgummel** setup. The setup inputs and outputs are displayed.

Table 16   For the vb input

Set	To
Mode	V (voltage)
Unit	SMU1
Compliance	no greater than the maximum allowable base current value in the device data sheet.
Sweep Type	LIN, to provide a linear Vbe sweep from start to stop voltage values
Start and Stop	Voltage values that define the operating range of the device. The Start value should be low enough so that the Ib curve bends up at the low end due to recombination in the base-emitter depletion region. The Stop value should be large enough so that the Ic curve begins to bend over at the high end due to high-level effects.
# of Points	A value sufficiently large such that the subtle bends at the low and high ends of the curves are clearly displayed—51 points is usually an adequate number

Table 17   For the ve input

Set	To
Mode	V (voltage)
Unit	GND
Sweep Type	CON
Value	0.000 to provide a constant zero voltage. This holds the emitter at ground potential to prevent reverse leakage from influencing Ib.

Table 18   For the vc input

Set	To
Mode	V (voltage)
Unit	SMU2
Compliance	no greater than the maximum collector current value in the data sheet.
Sweep Type	SYNC, to synchronize it to the Master Sweep vb.
Ratio	1.000
Offset	0.000

- The Unit names must conform to the names set in the hardware window. Refer to the appropriate chapter as follows:
    - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
    - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

They must also correspond with the actual plug-in SMU connections from the Agilent 4142.

- The Outputs are factory set, and should not be changed, with the exception of Type. This is the type of data to be plotted, M for measured, S for simulated, or B for both. The illustrations of measured data in this chapter were plotted with the outputs set to M: in the extraction section later in the chapter, where the illustrations show both measured and simulated data, the outputs were set to the default, B.

Measuring and Plotting

This procedure triggers the device measurement, controlling the DC source/monitor to supply the signals you defined in the instrument options and the inputs, and monitors the results.

NOTE	Remove any high-intensity light sources such as microscope light before taking a measurement. This is particularly important if currents in the nanoamp range are being measured. One simple method of blocking light is to place an opaque plastic box on the fixture directly above the device.

1 From the Plots tab, select the **ibicvsvb** plot, and **Display Plot**.

2 If you have not already connected the DUT, do so now. Observe antistatic precautions**.**

3 Select **Measure/Simulate > Measure**. *Do not touch* the fixture or the bias networks while the system is measuring. The yellow MEASURING light on the DC source/monitor will go off when the measurement is complete.

4 The plot will be updated with the measured data. If the Ic curve does not bend over at the high end, go back to the inputs and set vb Stop to a larger value. Be careful, however, not to exceed the maximum current rating of the device. If the Ib curve does not bend up at the low end, set vb Start to a smaller value. Repeat the measurement.

  The other plots are used later, in the extraction process.

5 Save the data using **File > Save As** and a filename with the suffix *.set*.

6 Close the plot.

Making the Reverse Gummel Measurement

This procedure measures Ie and Ib with respect to –Vce. The emitter and base are grounded, and the base-collector voltage is swept across a negative range wide enough to show both high-level effects and low-level depletion effects.

BJT Configuration: The input values you set in the following rgummel procedure configure the BJT as shown in the following figure.
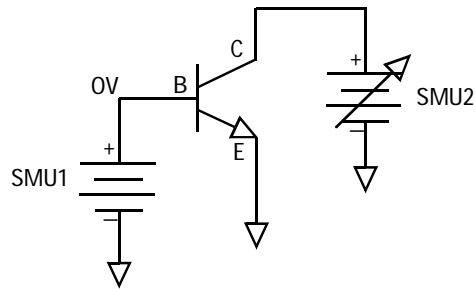
Figure 38   BJT Configuration for rgummel Measurement

Parameters

The data from this measurement will be used to extract BR, ISC, NR, IKR, and NC. It may also need to be used to interactively modify other parameters such as RE, RC, RB, and VAF.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Defining the DC Source/Monitor Instrument State" on page 621 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

Use the same settings for the rgummel setup as you did for the fgummel.

NOTE The instrument states need to be set independently for each setup.

Then return to this chapter and continue.

Setting the rgummel Input Values

1 In the DC DUT select the **rgummel** setup.

2 The vb Value is usually set to 0.000 to prevent reverse leakage from influencing Ib at low −Vc values.

3 Set the vc Start value low enough that the Ib curve preferably bends up at the low end due to recombination in the base-collector depletion region. The Stop value should be large enough that the Ie curve begins to bend over at the high end due to high-level effects. Synchronization is not necessary for this measurement.

4 Do not change the ve values, which set the emitter to ground.

Measuring and Plotting

1 From the Plots tab, select the **ibievsvc** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The plot will be updated with the measured data. If the Ie curve does not bend over at the high end, go back to the inputs and set vc Stop to a larger value. Be careful, however, not to exceed the maximum current rating of the device. Repeat the measurement.

3 Save the data using **File > Save As** and a filename with the suffix .set.

4 Close the plot.

Making the Forward Early Measurement

The forward early measurement models the effect of base-width modulation due to variations in the base-collector depletion region.

This procedure measures Ic with respect to Vce at a single value of Vbe. The emitter is grounded and the collector voltage is swept. The Vbe value is taken from the log-linear region of the fgummel measurement.

BJT Configuration: The input values you set in the following fearly procedure configure the BJT as shown in the following figure.

Figure 39   BJT Configuration for fearly Measurement

Parameters

The data from this measurement will be used to extract VAF, the forward early voltage.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Defining the DC Source/Monitor Instrument State" on page 621 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

Use the same settings as for the fgummel measurement, except set Integ Time to M (medium) if you prefer. Then return to this chapter and continue.

Setting the fearly Input Values

1 Select the **fearly** setup.

2 Set Vb to measure at a single constant value. This should be in the log-linear range of the fgummel measurement, between the high- and low-level effects. To determine the Vb value, refer to the measured data from the fgummel measurement.

3 The reason for measuring at only one value of Vbe is that the rearly measurement must use the same number of base voltage values. The rearly measurement uses the sync function, and it can only synchronize on one base value.

4 Set the vc range with a Start value of 0.0. Referring to the data sheet for your device, set the Stop value ≤ the upper limit of the device's normal operating range.

5 Set the vc # of Points sufficient that the resulting Ic curves will be clearly separated and easily distinguishable.

6 Do not change the ve values, which set the emitter to ground.

Measuring and Plotting

1 From the Plots tab, select the **icvsvc** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The plot will be updated with the measured data. If you are not satisfied with the data, go back to the inputs and change the vb or vc values, staying within the log-linear range of the fgummel measurement for vb. Repeat the measurement.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

4 Close the plot.

Making the Reverse Early Measurement

The reverse early measurement models the effect of base-width modulation due to variations in the base-emitter depletion region.

This procedure measures Ie with respect to –Vce. The base is synchronized to the collector. The Vbc values are taken from the –Vc values in the log-linear region of the rgummel measurement, taking the device from near cutoff to near saturation.

BJT Configuration: The input values you set in the following rearly procedure configure the BJT as shown in the following figure.
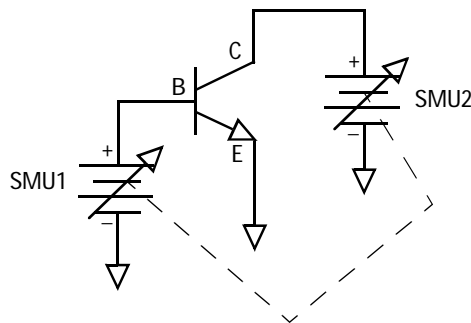
Figure 40   BJT Configuration for rearly Measurement

Parameters

The data from this measurement will be used to extract VAR, the reverse early voltage.

Defining the DC Source/Monitor Instrument State

Follow the same procedure you used in the fearly measurement. Use the same settings.

Setting the rearly Input Values

1 Select the **rearly** setup.

2 Set the vc range with a Start value of 0 and a Stop value in the negative range. This should be in the log-linear range of the rgummel measurement, between the high- and low-level effects.

3 Because vb is synchronized to the master sweep vc, only one vb value can be set. In this example, vb is offset from vc by 670 mV, so that Vb = Vc + 670 mV. The ratio of vb to vc is 1:1.

Measuring and Plotting

1 From the Plots tab, select the **ievsvc** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. If you are not satisfied with the measured data, go back to the inputs and change the vc values or the vb offset. Repeat the measurement.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the fearly_family Measurement

This setup measures Ic with respect to Vce at three values of Vbe. The collector is set to the same values as in the fearly measurement, but the base voltage is set for measurements at three values. This allows you to view the device behavior over a wider range of base input voltages.

BJT Configuration: The input values you set in the following fearly_family procedure configure the BJT as shown in the following figure.
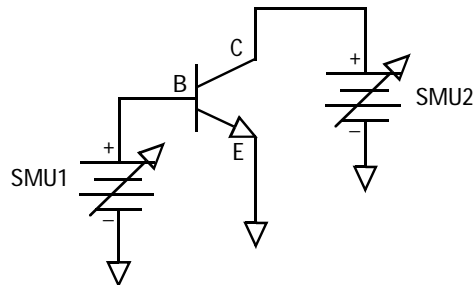
Figure 41   BJT Configuration for fearly_family Measurement

Defining the DC Source/Monitor Instrument State

Follow the same procedure you used in the fearly measurement. Use the same settings.

Setting the fearly_family Input Values

1 Select the **fearly_family** setup.

2 Set the vb # of Points to 3 to allow for measurements at three different values so that you can see the device behavior at different base voltages. One of the base voltage values is the same as the value used in the fearly measurement.

3 To determine the Vb values, refer to the measured data from the fgummel measurement. Set vb Start and Stop to values in the log-linear range of the fgummel measurement, between the high- and low-level effects. The step size should be sufficient that the resulting measured Ic curves will be clearly separated and easily distinguishable.

4 Set the vc range with Start and Stop values the same as for the fearly measurement.

Measuring and Plotting

1 From the Plots tab, select the **icvsvc** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. If you are not satisfied with the spacing of the Ic traces, go back to the inputs and change the vb Start and/or Stop values, staying within the log-linear range of the fgummel data. Repeat the measurement.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the rcflyback Measurement

This procedure drives the base with a current to negate the effect of the base parasitic resistance. The collector is synchronized to the base such that $-Ic = Ib$. The change in negative collector voltage is monitored vs the change in base current. The collector voltage is the base current times the parasitic resistance in the collector.

BJT Configuration: The input values you set in the following rcflyback procedure configure the BJT as shown in the following figure.
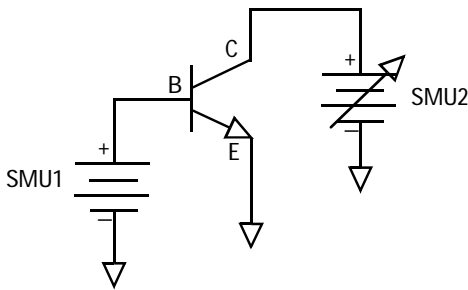
Figure 42   BJT Configuration for rcflyback Measurement

Parameters

The data from this measurement is used to extract the parasitic collector resistance RC.

Defining the DC Source/Monitor Instrument State

Follow the same procedure you used in the fearly measurement. Use the same settings.

Setting the rcflyback Input Values

1 Select the **rcflyback** setup.
2 In this setup, the ib inputs set the base current from the source/monitor to sweep from 100 uA to 10 mA, so that the measured data will be linear.
3 Set the ic Sweep Type to **SYNC**. The ic inputs synchronize the collector to the base, such that –Ic = Ib. This is done with a ratio of –1 and an offset of 0.

Measuring and Plotting

1 From the Plots tab, select the **vcvsib** plot, and **Display Plot**.
2 Select **Measure/Simulate > Measure**. The measured data should be similar to that shown in the following figure. The straight line is measured –Vc data. It should be fairly linear, and should only begin to deviate from a straight line at the

high end of the base current value, if at all. If necessary, you can go back to the inputs and change the ib Stop value, then repeat the measurement.
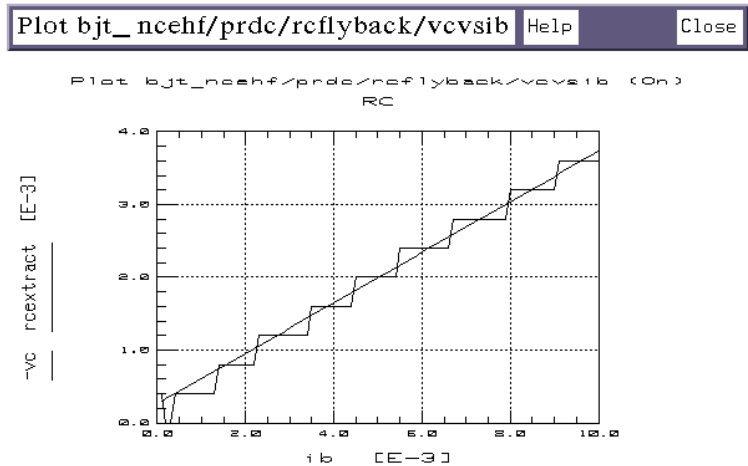
Figure 43   Example Measured rcflyback vcvsib Data

Making the reflyback Measurement

This procedure is similar to the rcflyback measurement. A current source drives the base to negate the effect of the base parasitic resistance. In this case the collector is open-circuited by setting its SMU current to zero. The change in collector voltage is monitored vs the change in base current. The collector voltage is the base current times the parasitic resistance in the emitter.

BJT Configuration: The input values you set in the following reflyback procedure configure the BJT as shown in the following figure.
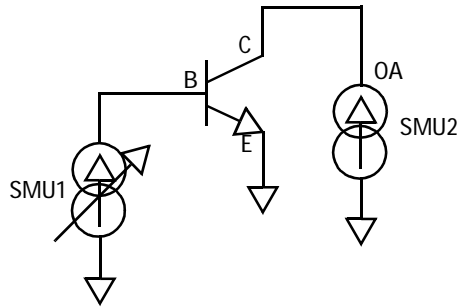
Figure 44 BJT Configuration for reflyback Measurement

Parameters

The data from this measurement is used to extract the parasitic emitter resistance RE.

Defining the DC Source/Monitor Instrument State

Follow the same procedure you used in the fearly measurement. Use the same settings.

Setting the reflyback Input Values

1 Select the **reflyback** setup.

2 Set the ib Start value to 0.000. Set Stop to a current value such that the measured Vc curve will be fairly linear. This is similar to the range used in the rcflyback measurement.

3 Set the ic Sweep Type to CON (constant) with a Value of 0.000, to open-circuit the collector.

Measuring and Plotting

1 From the Plots tab, select the **vcvsib** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The measured data should be similar to that shown in the following figure. Note that the stepped effect of the trace is caused by the resolution of the SMU in the DC source/monitor.
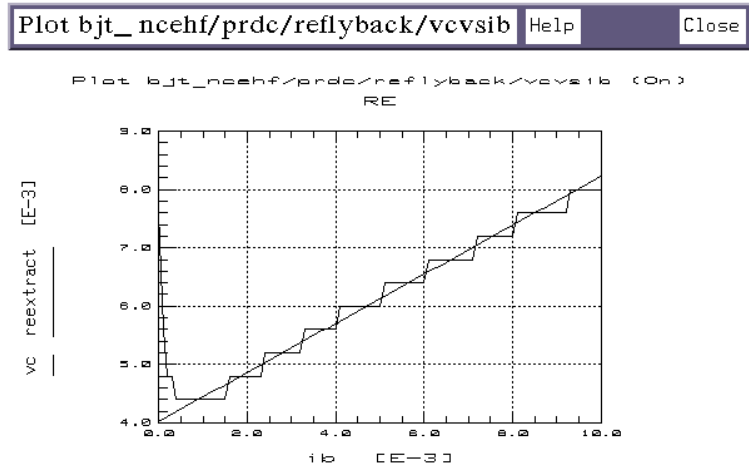
Figure 45   Example Measured reflyback vcvsib Data

3 The Vc trace should be fairly linear, and should only begin to deviate from a straight line at the high end of the base current value, if at all. If necessary, go back to the inputs and change the ib Stop value, then repeat the measurement.

Calibrating the Network Analyzer

It is important to calibrate the network analyzer before you perform any S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and extraction. Without a good calibration, the results of the extraction *will be* inaccurate.

The high-frequency BJT measurement procedure requires a swept broadband calibration and one or two CW calibrations. The network analyzer is calibrated over the operating range of the device for the swept-frequency S-parameter measurements used to extract the AC device parameters. In addition, a calibration is performed at a single frequency for the CW S-parameter measurements used to extract the parasitic capacitance parameters. If the measurement system you are using is Agilent 8510-based, the CW calibration can be a subset of the broadband calibration. For Agilent 8753- and 8720- based

systems, IC-CAP does not support the subset calibration technique, and separate standards measurements are needed for the swept and CW calibrations. In either case, a second CW calibration may be needed later for the h21vsvbe measurement.

CW Calibration Frequency

The frequency of the CW calibration for the capacitance measurements should be around 100 to 200 MHz to eliminate the effects of test set coupler roll-off at low frequencies, and low enough to eliminate bond inductance and other parasitic inductance considerations. With an Agilent 8510-based system, use a calibrated measurement point equal to the start frequency of the broadband cal plus an integer multiple of the step size: the frequency used in the example procedure is 110 MHz.

50 MHz (start freq) + 1 · 60 MHz = 110 MHz

The h21vsvbe measurement is also performed at a single CW frequency. However, this frequency is determined based on the data provided by the S-parameter measurements, and cannot yet be determined. When you reach the h21vsvbe measurement and determine the measurement frequency, you can use the same method to perform that calibration.

For step-by-step calibration procedures, refer to the appropriate chapter as follows:

- "Calibrating the Network Analyzer" on page 589 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (if you have an Agilent 8510-based system)

- "Calibrating the Network Analyzer" on page 631 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (if you have an Agilent 8753-based system)

Measuring the Junction Capacitances

The junction capacitances are measured with a calibrated network analyzer, rather than a CV meter, for the greatest measurement accuracy. Accurate capacitance measurements are extremely difficult at high frequencies, therefore the

parasitic capacitance values are derived directly from the S-parameters measured at the collector and emitter of the device, with a CW frequency signal applied to the base.

Setting up the measurement conditions involves processes similar to those for the DC measurements: defining the instrument states and setting the inputs.

Making the sparm_Cjunc Measurement

This procedure measures the base-emitter and base-collector junction capacitances. An RF signal at a CW frequency is applied to the base, while the base voltage is swept from slightly forward-biased to strongly reverse-biased. A network analyzer takes S-parameter measurements from which the junction capacitances are derived.

NOTE   This setup combines and replaces both the earlier sparm_cje and sparm_cjc setups. While this simplifies the measurement, the bias is limited to the maximum base-emitter reverse breakdown voltage. If necessary, you can create two junction setups, sparm_cje and sparm_cjc, and make two separate measurements.

BJT Configuration: The input values you set in the following sparm_Cjunc procedure configure the BJT as shown in the following figure.

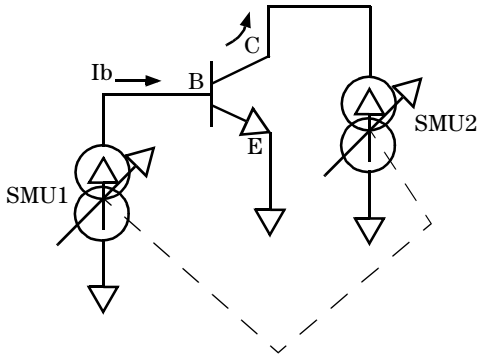**Figure 46**   BJT Configuration for sparm_Cjunc Measurement

Parameters

The data from this measurement is used to extract CJE, VJE, MJE, CJC, VJC, and MJC.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Instrument States for CW Frequency Measurements" on page 601 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Instrument States for CW Frequency Measurements" on page 628 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

Set the Cal Set No to the cal set or register where you stored your CW calibration. The procedures in this manual use cal set #2. Then return to this chapter and continue.

Setting the sparm_Cjunc Input Values

1 Connect the network analyzer RF input cables to the RF IN ports of the bias networks.

2 Select the **sparm_Cjunc** setup.

3 The swept base voltage biases both the base-emitter and the base-collector junctions with the same voltage. The applied bias must be limited to the maximum base-emitter reverse breakdown voltage to prevent damage to the device.

4 Set the vc Sweep Type to CON (constant) with a Value of 0.000, to open-circuit the collector.

5 Do not change the ve values, which set the emitter to ground.

6 Set the freq Sweep Type to CON (constant) to measure at a single CW frequency.

7 Set the freq Value to the frequency of your CW calibration (see "Calibrating the Network Analyzer," earlier). The frequency used in this example procedure is 110 MHz. The frequency should be in the MHz range to eliminate the effects

of test set roll-off at low frequencies, and low enough to eliminate bond inductance and other parasitic inductance considerations.

NOTE   In setting a MHz value, write *MEG* in full. IC-CAP is not case-sensitive, and assumes that *m* or *M* corresponds to *milli*. If the message **Error: NWA frequency value < 45 MHz** is displayed, you may have entered *M* only.

Measuring and Plotting

1 From the Plots tab, select the **cj_vs_v** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

4 Close the plot.

Measuring S-Parameters versus Frequency

The swept S-parameter measurements in the next two setups provide the data that allows extraction of parameters related specifically to the high-frequency performance of the device. The S-parameters are measured at the collector of the device, with a swept-frequency signal applied to the base. The first setup performs an S-parameter measurement without DC bias; the second is a DC-biased S-parameter measurement.

Making the sparm_0v Measurement

This procedure measures the S-parameters at the collector of the BJT across a calibrated microwave frequency range, with no DC bias signals applied.

BJT Configuration: The input values you set in the following sparm_0v procedure configure the BJT as shown in the following figure.
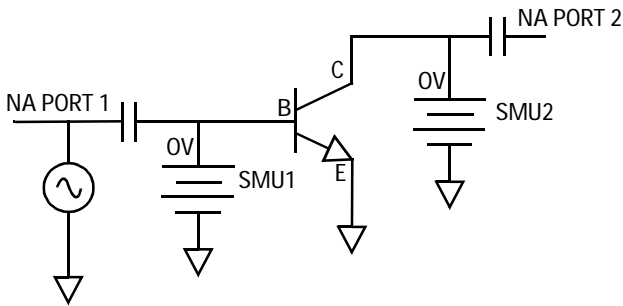
Figure 47   BJT Configuration for sparm_0v Measurement

Parameters

The data from this measurement will be used in the extraction of RB and LVIA.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Instrument States for Swept Measurements" on page 598 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Instrument States for Swept Frequency Measurements" on page 624 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (Agilent 8753- or 8720-based system)

Note: The instrument states need to be set independently for each setup, and are different for CW and swept frequency measurements. Set the Cal Set No to the cal set or register where you stored your broadband calibration. The procedures in this manual use cal set #1. Then return to this chapter and continue.

Setting the sparm_0v Input Values

1 Select the **sparm_0v** setup.

2 Set the Sweep Type for all the voltage inputs (vb, vc, and ve) to CON (constant), with a Value of 0.000, to apply zero bias at all terminals.

3 Set the freq Sweep Type to LIN, for a linear measurement sweep across the frequency range. Set the Start and Stop values to correspond with the frequency range you defined in your broadband calibration. The frequency range used in the example procedure is 50 MHz to 6.05 GHz. Set the # of Points the same as in the calibration.

Measuring and Plotting

1 From the Plots tab, select the **s11** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**. The measured data should be similar to that shown in the following figure, though the trace may or may not cross the real (horizontal) axis.
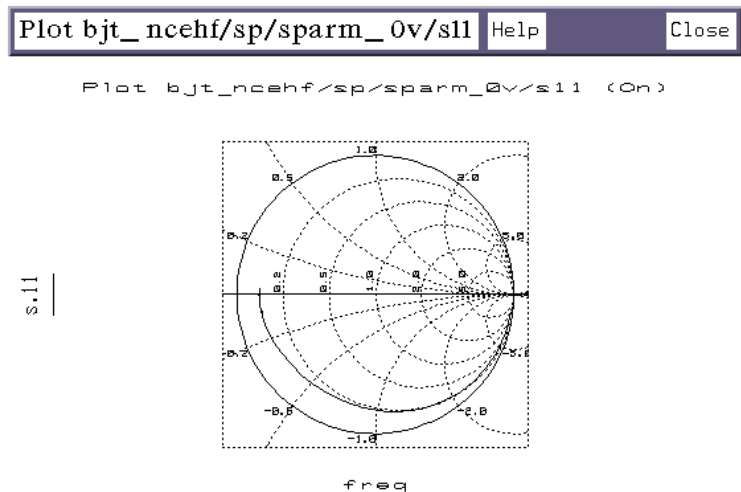
Figure 48   Example Measured sparm_0v s11 Data

The following steps determine the device resonant frequency F_r (the frequency where the S11 curve crosses the real axis), and the input impedance $R(@F_r)$.

1 If the trace does not cross the real axis, you have two alternatives:

- You can increase the network analyzer upper frequency limit until the trace does cross the real axis (a time-consuming solution because you will need to recalibrate).

- Or (preferably) you can visually extrapolate the S11 trace to see where it would cross the real axis if it were actually extended.

2 After you find the point where the trace crosses (or would cross) the axis, on the network analyzer press LOCAL to obtain front-panel control. Then press PARAMETER > S11 > FORMAT > SMITH CHART > MARKER > MARKER 1. Turn the RPG knob to move the marker to the point where the trace crosses the real axis. The marker readout shows the frequency at that point, as well as the real and imaginary impedance values.

3 Record the frequency: this is the resonant frequency F_r of the device.

4 Record the real impedance value: this is the input impedance $R(@F_r)$.

5 Save the measured data using **File > Save As** and a filename with the suffix *.set*.

Making the sparm_bias Measurement

This procedure measures the S-parameters at the collector of the BJT across a calibrated microwave frequency range, at three values of base-emitter bias that take the device from near cutoff to near saturation.

BJT Configuration: The input values you set in the following sparm_bias procedure configure the BJT as shown in the following figure.
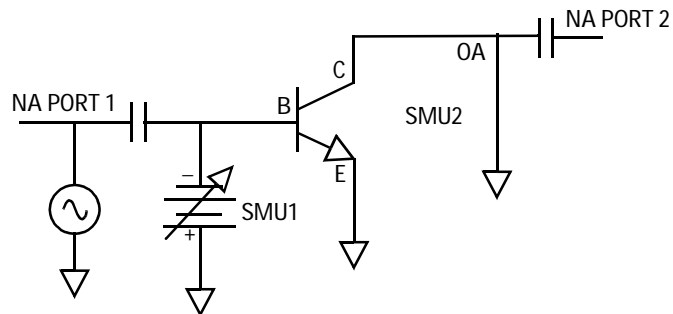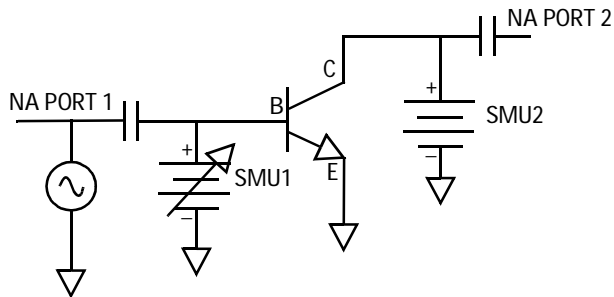
Figure 49   BJT Configuration for sparm_bias Measurement

Parameters

The data from this measurement will be used in the extraction of RB, IRB, RBM, TF, PTF, FC, and XCJC. This data will also be used to check the overall quality of all the extracted parameters.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Instrument States for Swept Measurements" on page 598 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Instrument States for Swept Frequency Measurements" on page 624 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (Agilent 8753- or 8720-based system)

Use the same settings as you did for the sparm_0v setup. Then return to this chapter and continue.

Setting the sparm_bias Input Values

1 Select the **sparm_bias** setup.

2 Set the vb # of Points to 3, to provide three different bias levels.

3 To determine the Vb values, refer to the measured data from the fgummel measurement. Set vb Start to a value near device cutoff, and Stop to a value near saturation.

4 Set the vc Sweep Type to CON (constant), with a Value at or near the normal operating point of the device.

5 Do not change the ve values.

6 Set the freq values the same as you did for the sparm_0v setup, to correspond with your broadband calibration.

Measuring and Plotting

1 From the Plots tab, select **Display All** to display all the S-parameter plots.

2 Select **Measure/Simulate > Measure**. Note that the db_h21 plot shows the device gain H21 at the three bias levels.

3 If the db_h21 plot shows an unexpected drop in the gain of the device with bias (an abrupt change of several dB between one measurement point and the next), it is caused by thermal runaway. In this case, go back to the inputs and reduce the collector voltage by changing the vc Value. Then repeat the measurement.

4 When you are satisfied with the measured data, save it using **File > Save As** and a filename with the suffix *.set*.

Making the h21vsvbe Measurement

This procedure uses the network analyzer to measure H21 (the device current gain) versus Vbe, at a single microwave frequency. The base voltage is swept from near cutoff to near saturation. The H21 magnitude is derived from the S-parameters measured at the collector.

The measurement frequency is determined from the sparm_bias measurement data. A frequency is chosen in the linear range of the db_h21 plot. A network analyzer calibration is performed at that frequency prior to the h21vsvbe measurement, unless the frequency is the same as the frequency used for the sparm_Cjunc measurement.

Because this measurement data will be combined with the data from the ftic measurement performed next, the Vb and Vc values used in both measurements need to be identical. This is accomplished by setting input values for both procedures to variables that are defined in the model variables table, as explained below.

BJT Configuration: The input values and measurement variables you set in the following procedures configure the BJT as shown in the following figure.
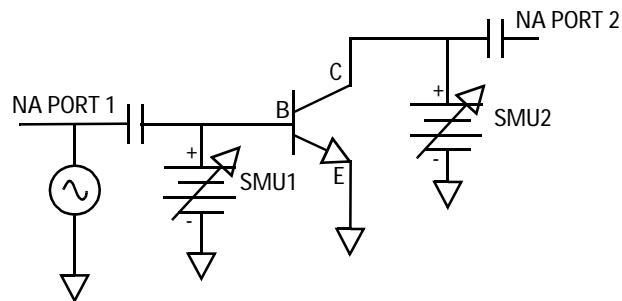
Figure 50   BJT Configuration for h21vsvbe Measurement

Parameters

The data from this measurement and the ftic measurement will be used to generate Ft vs Ic. This will then be transformed to extract TF, VTF, XTF, and ITF.

Determining the Measurement Frequency

The frequency value for this measurement is determined from the sparm_bias measurement data.

1 Referring to the db_h21 plot from your sparm_bias measurement, note that part of the trace has a roll-off of approximately 20 dB per decade. Within this portion of the trace, select a frequency near the low end (see Figure 51).

2 To take advantage of the easy calibration subset capability of an Agilent 8510-based system, choose a frequency equal to the broadband cal start frequency plus an integer multiple of the calibration step size. Typically a frequency close to

200 MHz is adequate. The frequency used in the example procedure is 170 MHz.

50 MHz (start freq) + 1 · 120 MHz = 170 MHz

(If you have an Agilent 8753- or 8720-based system, you cannot use the cal subset feature with IC-CAP. Choose any appropriate frequency in the 20 dB per decade region of the trace.)
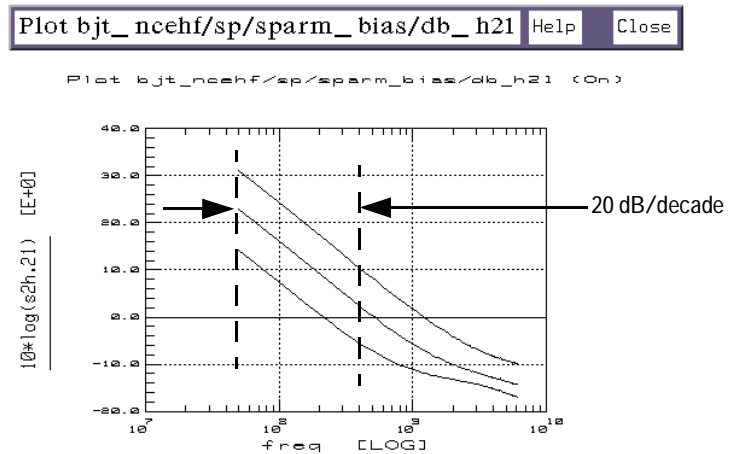
Figure 51   Determining the Frequency for the h21vsvbe Measurement

Calibrating the Network Analyzer

NOTE	If the frequency you selected for this measurement is the same as the frequency of the sparm_Cjunc measurement, you need not perform another calibration.

If the frequency you selected is not the same, and your system is Agilent 8510-based, perform the following steps to create a CW frequency calibration subset of the original broadband cal:

1 On the Agilent 8510 network analyzer, press CAL > CORRECTION ON > CAL SET 1 to turn on the broadband

calibration. Then press MORE > MODIFY CAL SET > FREQUENCY SUBSET.

2 Use the SUBSET: START and SUBSET: STOP softkeys in the subset menu (*NOT* the front panel keys) to set both start and stop to the same frequency.

3 Press CREATE & SAVE, and store this calibration in cal set #3.

If your system is Agilent 8753- or 8720-based and the frequency you selected is not the same as the frequency of the sparm_Cjunc measurement, perform a new CW calibration at that frequency.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

• "Instrument States for CW Frequency Measurements" on page 601 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

• "Instrument States for CW Frequency Measurements" on page 628 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

Use the same settings as you did for sparm_0v and sparm_bias, unless you performed a new calibration for this measurement, set Cal Set No to 3. Then return to this chapter and continue.

Setting the h21vsvbe Input Values

1 Select the **h21vsvbe** setup.

2 Under vb, set Start to h21start, set Stop to h21stop, and set # of Points to h21npts. These are set to variables, instead of numerical values, to ensure the values will be identical to those used in the ftic measurement. The method for setting the actual values is described in the section, "Defining the h21 Measurement Variables" on page 150.

3 Under vc, set Start to h21stop, so that the Vc start value equals the Vb stop value. Set Stop to a value close to the
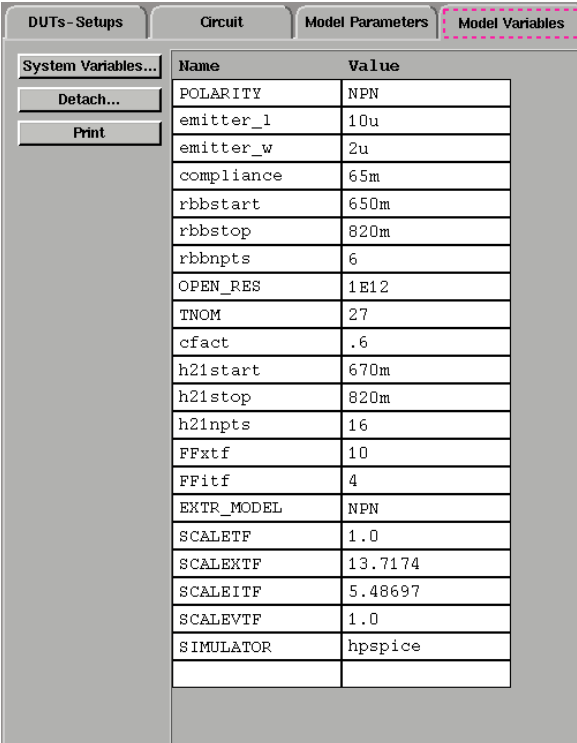
normal Vc operating point. However, if heat dissipation of the device fixture is a concern, keep the collector bias below 2V.

4   Set the vc # of Points to 2, to measure at the lower and upper Vc values.

5   Do not change the ve values.

6   Set the freq Sweep Type to CONstant to measure at a single frequency. Set the Value to the frequency you defined at the beginning of this procedure.

Defining the h21 Measurement Variables

This assigns numerical values to the variables listed as inputs. (The same values will apply to the ftic measurement.)

1   Select the **Model Variables** tab. The model variable table is displayed, as shown in the illustration.

Name	Value
POLARITY	NPN
emitter_l	10u
emitter_w	2u
compliance	65m
rbbstart	650m
rbbstop	820m
rbbnpts	6
OPEN_RES	1E12
TNOM	27
cfact	.6
h21start	670m
h21stop	820m
h21npts	16
FFxtf	10
FFitf	4
EXTR_MODEL	NPN
SCALETF	1.0
SCALEXTF	13.7174
SCALEITF	5.48697
SCALEVTF	1.0
SIMULATOR	hpspice

Tabs: DUTs-Setups | Circuit | Model Parameters | Model Variables

Buttons: System Variables... | Detach... | Print

2 Set h21start and h21stop to appropriate Vbe values that will take the device from near cutoff to near saturation. The lower Vbe limit should not be so low that the device is cut off. The upper limit should be high enough that the measured H21 curve will begin to roll-off due to higher order effects. Typical values range from Vbe = 0.65V to 0.8V.

3 Set h21npts (number of points) to a value that will make the voltage step size equal to an integer multiple of the Agilent 4142 voltage resolution. For information on the voltage resolution of Agilent 4142 SMUs, refer to the "Getting Started" chapter of the Agilent 4142B *Operation Manual*.

Measuring and Plotting

1 Make the DUT/Setups folders active again and select the **Plots** tab. Select the **h21_vbe** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**.

3 If the measured data shows an unexpected drop in magnitude (an abrupt change of several dB between one measurement point and the next), it is caused by thermal runaway. The following figure shows an example. In this case, go back to the variable table and reduce the upper limit of Vbe by reducing the h21stop value. Then repeat the measurement.
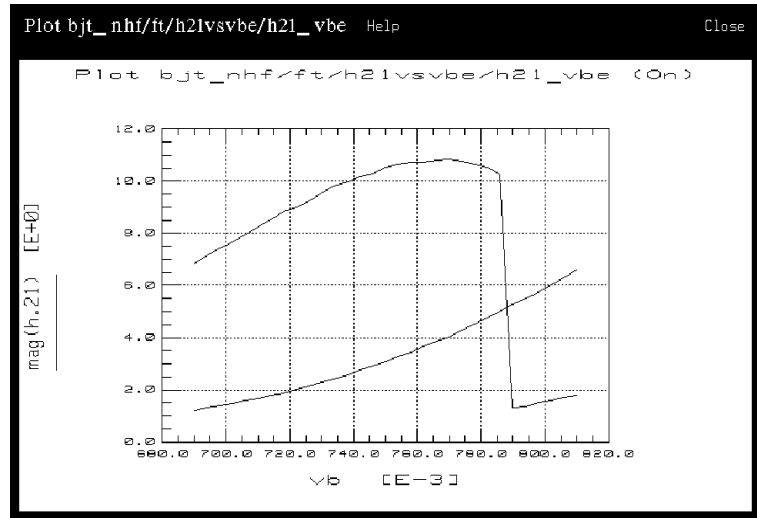
Figure 52   Effects of Thermal Runaway

4 When you are satisfied with the measured data, save it using
**File > Save As** and a filename with the suffix *.set*.

Making the ftic Measurement

This is a DC-only measurement that monitors the change in
collector current with respect to a change in base-emitter
voltage. The device configuration is the same as for the
h21vsvbe measurement, except there is no RF input signal.

Because this measurement data will be combined with the data
from the h21vsvbe measurement performed previously, the Vb
and Vc values used in both measurements need to be identical.
For this reason, some of the input values are set as variables
that were defined in the h21vsvbe procedure.

NOTE   The network analyzer test port cables can remain connected.

BJT Configuration: The input values you set in the following ftic procedure configure the BJT as shown in the following figure.
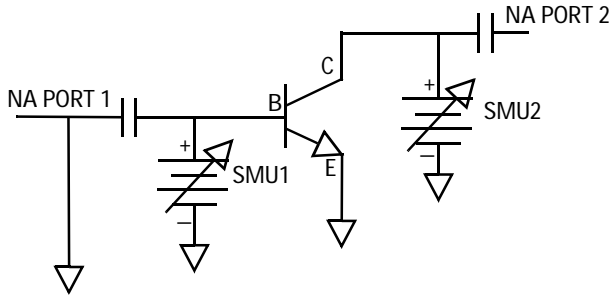
Figure 53   BJT Configuration for ftic Measurement

Parameters

The data from this measurement and the h21vsvbe measurement is used to generate Ft vs Ic. This is then transformed to extract TF, VTF, XTF, and ITF.

Defining the Instrument States

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System" (for an Agilent 8510-based system)

- "Defining the DC Source/Monitor Instrument State" on page 621 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" (for an Agilent 8753- or 8720-based system)

Use the same settings as you did for the h21vsvbe setup.

Setting the ftic Input Values

1 Select the **ftic** setup.

2 Under vb, set Start to `h21start`, set Stop to `h21stop`, and set # of Points to `h21npts`, to ensure these values are identical to those used in the h21vsvbe measurement.

3 Set vc Start to `h21stop`, so that the Vc start value equals the Vb stop value.

4 Set vc Stop to the same value as in the h21vsvbe measurement.

5 Set the vc # of Points to 2.

6 Do not change the ve values.

Measuring and Plotting

1 From the Plots tab, select the **ft_ic** plot, and **Display Plot**.

2 Select **Measure/Simulate > Measure**.

3 When you are satisfied with the measured data, save it using **File > Save As** and a filename with the suffix *.set*.

4 Close the plot.

Parameter Extraction

The extraction procedures are presented in the appropriate sequence to allow earlier extracted parameters to be used as a basis for later extractions.

In a BJT model, the parameters are very interdependent because of the inherent coupling of the model equations. The purpose of the different setups in the measurement process is to decouple the SPICE equations as much as possible. In the extraction process, the effects of individual parameters or small groups of related parameters can be isolated, and their values determined with reasonable certainty. The goal is to create a list of realistic parameters that can be used in a model to accurately predict device performance. Because of the interdependence of the parameters, the extraction process is extremely iterative, and involves many subjective judgments. The validity of the extracted parameters must be checked repeatedly.

The ability to discern reasonable values from questionable ones comes from knowledge of device physics, familiarity with a particular device, and individual experience. Parameter extraction is very much an art.

Displaying the Parameters Table

The bjt_ncehf Model Parameters table lists all the parameters that will be extracted, and initially displays default values. As you perform the individual extractions, the default values change to extracted values. Throughout the extraction process, you will use the Parameters table to read extracted values, and to enter or modify values to adjust interactive parameters for best agreement of simulated with measured data.

1 Click the **Model Parameters** tab. The table of model parameters is displayed, showing the current values of the individual parameters. The illustration shows a portion of the table.

Param Name	Value	Min	Max
LE	101.5p		
LB	195.9p		
CC	250.5f		
Q1.AREA	1.000		
NPN.IS	1.467f		
NPN.BF	221.5		
NPN.NF	991.5m		
NPN.VAF	44.63		
NPN.IKF	100.0		
NPN.ISE	282.3f		
NPN.NE	2.500		
NPN.BR	6.493		
NPN.NR	990.2m		
NPN.VAR	1.841		
NPN.IKR	100.0		
NPN.ISC	5.706f		
NPN.NC	1.171		
NPN.RB	3.579		
NPN.IRB	1.000p		
NPN.RBM	3.202		
NPN.RE	421.1m		
NPN.RC	349.2m		

Figure 54 Top Portion of the Model Parameters Table

2 To save the Parameters table after your device parameters are extracted, select **File > Save As**.

3 In the dialog box, select the File Type **.mps** for model parameter set.

4 Type a filename or use the one in the File Name field and click **OK**.

You can now recall the same set of device parameters any time you wish.

Extracting the Junction Capacitance Parameters

This procedure extracts the base-emitter parameters CJE, VJE, and MJE; and the base-collector parameters CJC, MJC, and VJC. These parameters are extracted from the data measured in the sparm_Cjunc setup.

Parameter	Description
CJE	Zero-bias base-emitter capacitance.
VJE	Base-emitter junction built-in potential.
MJE	Base-emitter grading coefficient.
CJC	Zero-bias base-collector capacitance.
VJC	Base-collector junction built-in potential.
MJC	Base-collector grading coefficient.

1 Scroll the Parameters table until it shows the values for CJE, VJE, and MJE.

2 Select **DUT/Setups > sparm_Cjunc**.

3 From the Plots tab, select the **cj_vs_v** plot, and **Display Plot**.
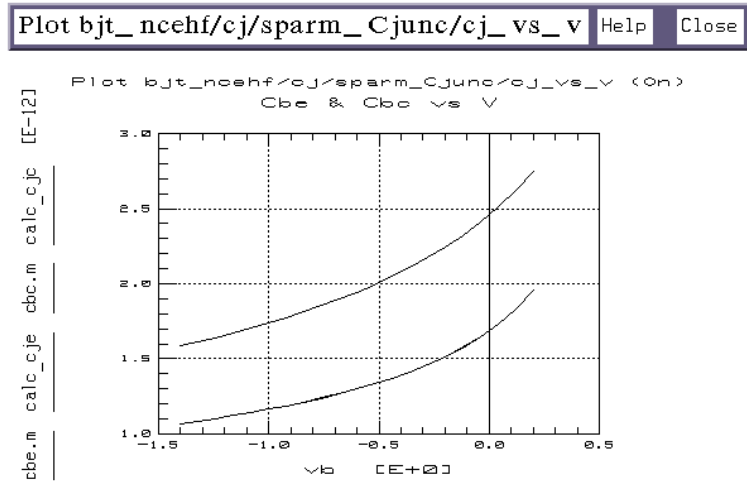
Figure 55 Measured and Simulated sparm_Cjunc cj_vs_v Data

4 Select the **Extract/Optimize** tab.

5 Select the **set_Cjunc** transform, and **Execute**. The transform calculates the values for CJE and CJC and enters them in the Parameters table.

6 Select **calc_cje > Execute**. Then select **opt_cje > Execute**. The simulated cje data in the plot should converge on the measured data, as in the previous figure.

7 Select **calc_cjc > Execute**. Then select **opt_cjc > Execute**. The simulated cjc data in the plot should converge on the measured data.

8 Close the plot.

Extracting the Collector Parasitic Resistance RC

This procedure extracts RC, using the data measured with the rcflyback setup. The measurement drives the base with a current source to negate the effects of the base parasitic resistance, and monitors −Vc vs Ib with Ie at 0 amps. The measured value at Vc is therefore equal to Ib times RC.

1 Select the **rcflyback** setup.

2 Select **Plots > vcvsib > Display Plot**. The plotted −Vc voltage is the product of the base current times the collector parasitic resistance (see the following figure).
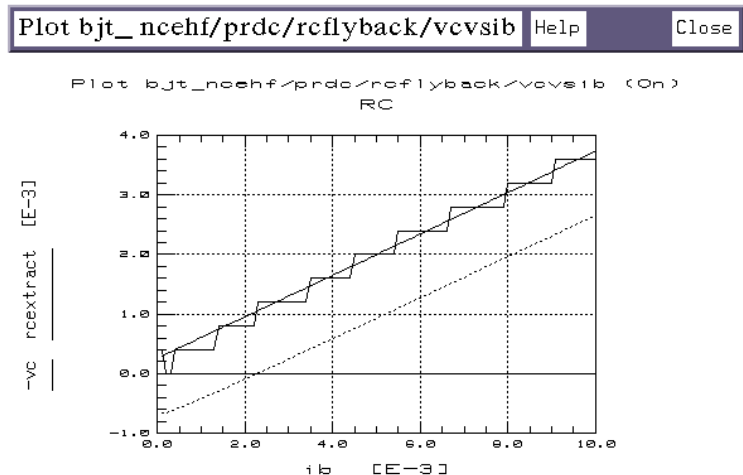
Figure 56   Example Plotted rcflyback vcvsib Data

3 Select the **Extract/Optimize** tab.

4 Select the **rcextract** transform, and **Execute**. The transform calculates RC as the slope of the −Vc vs Ib curve, and enters the value in the Parameters table.

5 Close the plot.

Extracting the Emitter Parasitic Resistance RE

This procedure is similar to the previous one. It extracts RE, using the data measured with the reflyback setup. The measurement drives the base with a current source to negate the effects of the base parasitic resistance, and monitors Vc vs Ib with Ic at 0 amps. The measured value at Vc is therefore equal to Ib times RE.

1 Select the **reflyback** setup.

2 Select **Plots > vcvsib > Display Plot**. The plotted collector voltage is the product of the base current times the emitter parasitic resistance (see the following figure).
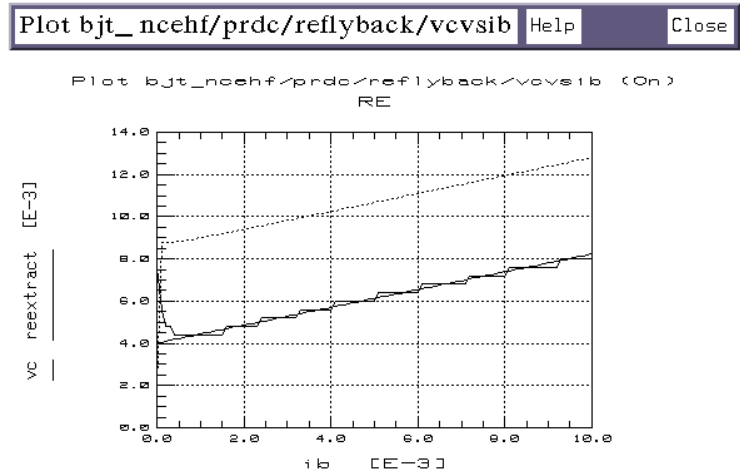
Figure 57   Example Plotted reflyback vcvsib Data

3 Select **Extract/Optimize > reextract > Execute**. The transform calculates RE as the slope of the Vc vs Ib curve, and enters the value in the Parameters table.

4 Close the plot.

The Intrinsic and Extrinsic Inductances

The intrinsic inductances of high-frequency BJTs are usually relatively small. At the frequencies where a BJT is normally used, they have essentially no effect on the device model. Compared to extrinsic parasitic inductance values such as the bond inductances, the intrinsic inductance values are small enough that they can be ignored.

A greater effect is caused by the extrinsic bond inductances LB and LE. In addition, if a carrier is used that has a via hole connecting the emitter pad to the backside ground plane, an additional amount of inductance is inherent in the via. This parasitic inductance is LVIA: its value is added to the value of LE.

The next procedure records the values for LB and LE; the following procedure extracts LVIA where applicable.

The Bond Inductances LB and LE

For this step it is necessary that the base and emitter bond inductances of your device are known. For example, if you know the inductance per unit length of the bond wire used, you can calculate the total bond wire inductance. Be sure to minimize the bond inductances, by making the bond length as small as possible or the bond wire as thick as possible.

1 Scroll the Parameters table until it shows the parameters LB and LE.

2 Manually enter the base and emitter bond inductance values for your device.

Extracting LVIA and RB

In a device mounted in a carrier that has a via hole connecting the emitter pad to the backside ground plane, LVIA is the inductance of the via. RB is the zero-bias base resistance.

NOTE   If you are making on-wafer probed measurements rather than in-fixture measurements, you can ignore the effects of LVIA (but not of RB).

This procedure uses data from the sparm_0v measurement to solve two equations. At zero bias the input of a BJT can be modeled as a series RLC circuit, where:

R = RE + RB
L = LE + LB + LVIA
C = CJE0

(Note that this ignores the effects of CJC0 and the BJT output circuit. The resultant error for a typical high-frequency device with an f_T of >4 GHz is <10%.)

1 From the sparm_0v measurement, note the values you recorded for the device resonant frequency F_r and the input impedance $R(@F_r)$.

2 Refer to the Parameters table for the value of RE you extracted earlier.

3 Use a calculator to find the value of RB by solving this equation:

$RB = R(@F_r) - RE$

4 Manually enter the calculated value for RB in the Parameters table.

5 If applicable, use the previously recorded values for LB and LE to find the value of LVIA by solving this equation:

$LVIA = [CJE \cdot (2 \cdot \pi \cdot F_r)^2]^{-1} - (LB + LE)$

6 Add the value of LVIA to the value of LE in the Parameters table, and manually enter the total into the Parameters table as the new value for LE.

Extracting VAR and VAF

This procedure is very iterative. The values for both VAR (the reverse early voltage) and VAF (the forward early voltage) are strongly dependent on the saturation current IS. The value of IS, however, is in turn strongly dependent on VAR. Therefore, the procedure begins with a first-pass rough extraction of IS, which is used in the extractions of VAR and VAF. After the values of VAR and VAF are extracted and optimized, IS will be extracted again using these values, in the next procedure.

First Estimate of IS

1 Select the **fgummel** setup.

2 Do not open any plots.

3 Select **Extract/Optimize > isextract > Execute**. The transform calculates a first estimate value for IS and enters it in the Parameters table.

Extracting VAR

1 Select the **rearly** setup.

2 Select **Plots > ievsvc > Display Plot**. The plotted data should be similar to Figure 59.

3 Select **Extract/Optimize > evextract > Execute**. This calculates a first estimate of both VAR and VAF.

4 The VAR extraction will be optimized over only the linear portion of the Ie trace. The next steps use the X-low/X-high function to select a portion of the trace.

  • Select the **Setup Variables** tab. A variable table will appear, listing X_LOW and X_HIGH with their current values, as shown in the illustration.

Instrument Options	Setup Variables	Extrac
Name	**Value**	
X_LOW	211.1m	
X_HIGH	914.4m	

Figure 58   The rearly Setup Variables Table

  • From the plotted data choose a linear portion of the Ie trace, along the constant slope of the saturated region (as illustrated by the box in Figure 59). Avoid any curved area at the high end of the trace. Click the mouse pointer at the lower and upper boundary points of the linear portion to make a box on the trace.

  • From the plot menu select **Options > Copy to Variables**. This changes the X_LOW and X_HIGH values for this plot to your boundary points.

5 Select **Extract/Optimize > opt_var > Execute**. The transform converges the simulated data to the measured data in the plot. It calculates the value of VAR using the linear portion of the Ie trace you defined with X-low/X-high, and enters the value in the Parameters table.
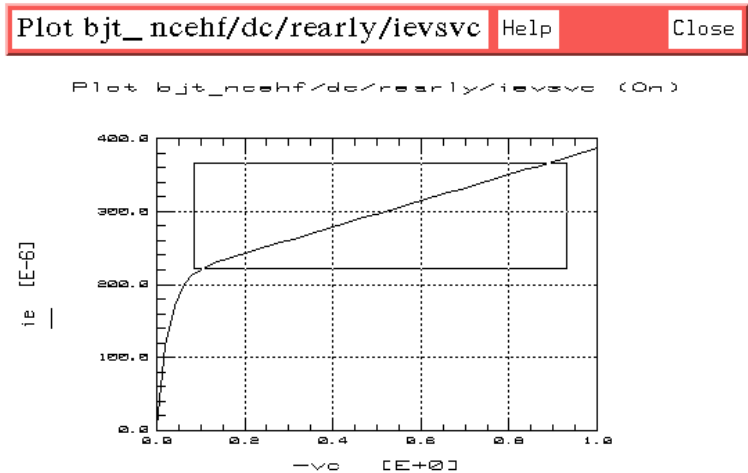
Figure 59   Plotted rearly ievsvc Data for VAR Extraction

Extracting VAF

This is similar to the procedure for VAR, using the fearly setup.

1 Select the **fearly** setup.

2 Select **Plots > icvsvc > Display Plot**.

3 Use the X-low/X-high function to define a linear portion of the Ic trace for the extraction, as explained under "Extracting VAR" on page 163.

4 Select **Extract/Optimize > opt_vaf > Execute**. The transform converges the simulated data to the measured data. It calculates the value of VAF using the portion of the trace you defined, and enters the value in the Parameters table.

5 Close the plot.

Extracting IS, NF, BF, ISE, NE, and IKF

This procedure uses several different transforms that isolate and optimize the individual parameters in the regions where they have the most dominant effect. These forward parameters are all extracted using the data measured with the fgummel setup. Note that extractions based on the fgummel measurement make the assumption that $Ic = -(Ie + Ib)$.

Table 19   Forward parameters extracted using the fgummel setup

Parameter	Description
IS	Transport saturation current.
NF	Forward current emission coefficient (models the deviation of the emitter-base diode junction from the ideal).
BF	Ideal maximum forward beta.
ISE	Base-emitter leakage saturation current (the value of the base current at zero base-emitter voltage).
NE	Base-emitter leakage emission coefficient (models the recombination of the base current in the base-emitter space charge region at low base-emitter voltage).
IKF	Forward knee current (models the drop in forward beta at high collector currents due to high-level injection).

Initial Extraction

1 Select the **fgummel** setup.

2 Select **Extract/Optimize > isextract > Execute** to extract initial values for IS and NF.

3 Select **fgextract > Execute** to extract initial values for IKF, BF, ISE, and NE.

4 Select **Plots > ibicvsvb > Display Plot**, and **nf_vs_vb > Display Plot**.

5 Select **Measure/Simulate > Simulate**.

6 Observe the plots. The simulated data should be reasonably close to the measured data. If so, proceed to "Optimizing IS and NF."

7 If the simulated data does not match fairly well with the measured data, the optimizer may not converge properly. In this case, extract the parameters *manually* with the Draw Diag Line feature, using the "Manual Extractions" explanations at the end of this procedure. Then come back to the optimization procedures given below.

Optimizing IS and NF

This optimization is performed over a section of the Ic curve at medium Vbe values, avoiding the low and high current effects.

1 Using the ibicvsvb plot (Figure 60), choose a section of the Ic (upper) curve at medium Vbe values. Avoid the low and high current effects. Alternatively, use the nf_vs_vb plot and choose a Vbe range where NF is reasonably flat and near minimum (see Figure 61).
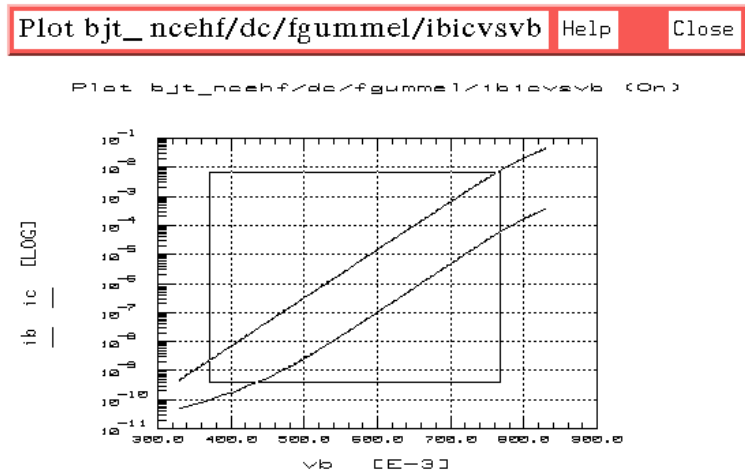
Figure 60   Appropriate Region of Ic for Optimization of IS and NF

2 Use the X-low/X-high function to define the section of the curve to be optimized:

   • Position the pointer at the lower end of your chosen range and click left, then at the high end and click left.

- From the plot menu select **Options > Copy to Variables**, to change the X-low/X-high values to the trace limits you selected.

3 Select **Extract/Optimize > optic_is_nf > Execute**. The transform converges the simulated data to the measured data in the plot. It optimizes the values for IS and NF using the section of the curve you defined, and enters the values in the Parameters table.
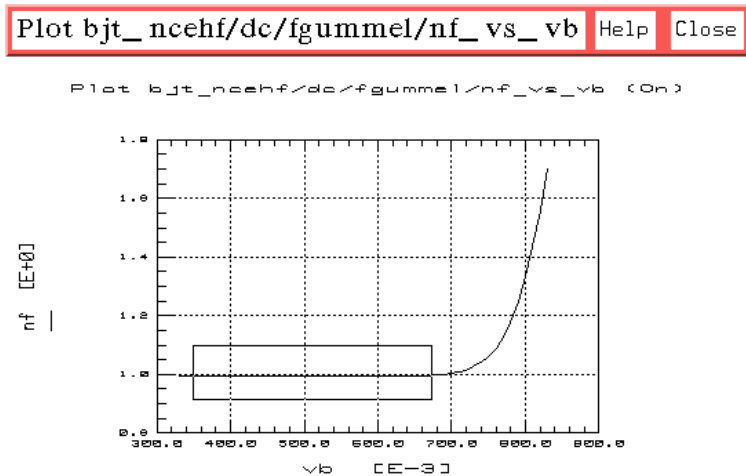
Figure 61   Appropriate NF Region to Optimize IS and NF

Optimizing BF

1 Use the same linear section of the Ic curve as in "Optimizing IS and NF" on page 166.

2 Select **opt_beta > Execute**. The transform optimizes the value for BF using the defined section of the Ic curve, and enters the value in the Parameters table.

Optimizing ISE and NE

This optimization is performed at the low end of the Ib curve.

1 Using the ibicvsvb plot, choose a section at the low end of the Ib curve where the slight bending upward of the trace

indicates the recombination in the base-emitter depletion region. Refer to Figure 62: the portion of the trace in the box is the appropriate region for optimization.
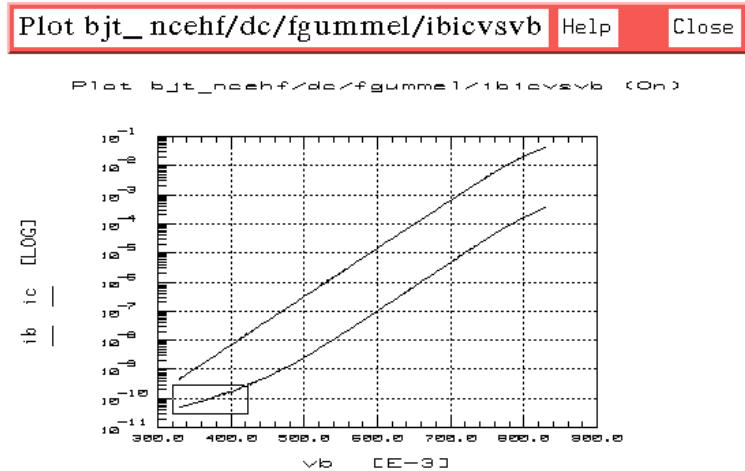
Figure 62   Appropriate Region of Ib for Optimization of ISE and NE

2 Use the X-low/X-high function to define the section of Ib to be optimized.

3 Select **optib_ise_ne > Execute**. The transform converges the simulated data to the measured, and enters the optimized values for ISE and NE in the Parameters table.

Optimizing IKF

This optimization is performed at the high end of the Ib and Ic curves.

1 Using the ibicvsvb plot, choose a section at the high end of both curves where they begin to deviate from a straight line. Refer to Figure 63: the portion of the traces in the marked box is the appropriate region for optimization. Alternatively, look at the nf_vs_vb plot (Figure 61), and set the lower limit for optimization above the point where NF begins to deviate

from a straight line. The upper limits for Ic and Ib are set by
the maximum value of Vb.
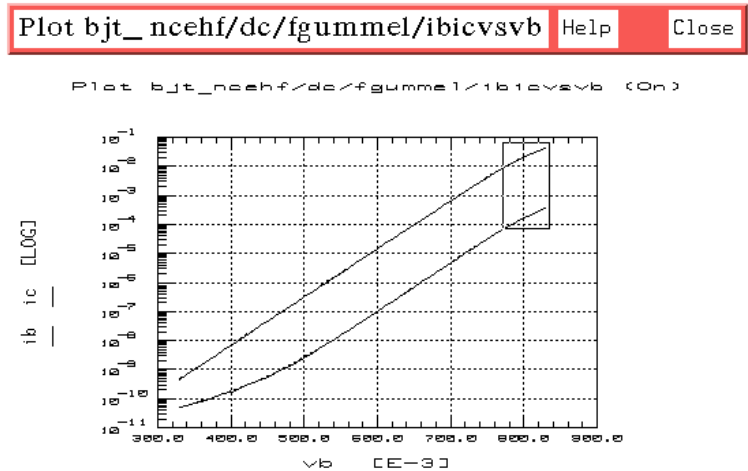
Figure 63   Appropriate Region of Ib and Ic for Optimization of IKF

2 Use the X-low/X-high function on one of the traces, to define
the range to be optimized.

3 Select **opt_ikf > Execute**. The transform converges the
simulated data to the measured, and enters the optimized
value for IKF in the Parameters table.

Manual Extractions (Optional)

Use these procedures only if the data simulated in "Initial
Extraction" on page 165 does not match reasonably well with
the measured data.

Manually Extracting IS and NF

1 Use the ibicvsvb plot. Choose a linear portion of the Ic vs Vb
curve, avoiding the low and high current effects. Put the
mouse pointer at the low end of the chosen portion and click

left, then put the pointer at the high end and click left, to make a box on the trace.

2 From the plot menu select **Options > Draw Diag Line**. A diagonal line is drawn on the plot that extends the linear portion you chose to the boundaries of the plot (see the following figure for an example). Annotations below the plot give the values for:

Slope of the diagonal line
Yo, the y-axis intercept of the diagonal line
Xo, the x-axis intercept of the diagonal line

3 Manually enter the Yo value into the Parameters table as the value for IS.
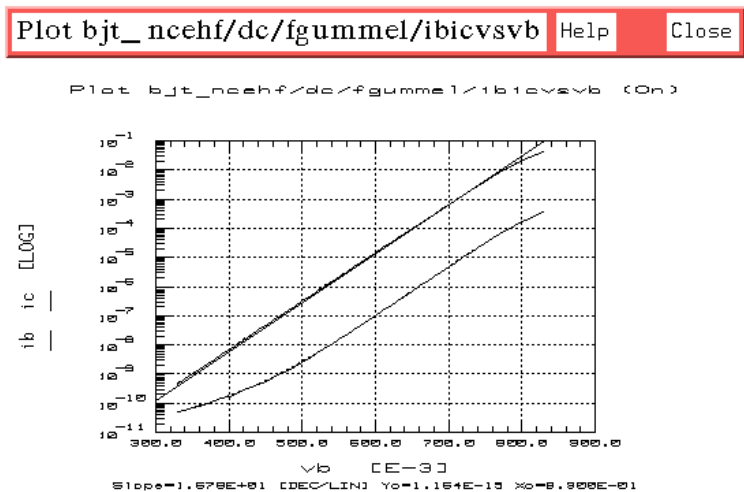
Figure 64 Manual Extraction of IS and NF Using Draw Diag Line

4 Solve the following equation (using a calculator) to determine the value of NF. Note that this equation is only valid up to high injection effects.

$$NF = \frac{1}{V_T}\left[\frac{d \cdot log_e(Ic)}{d \cdot Vbe} - \frac{1}{VAR - VBE}\right]^{-1}$$

where: $V_T = \dfrac{kT}{q}$

and:

elementary charge q   = $1.60218 \cdot 10^{-19}$

Planck's constant k   = $1.38066 \cdot 10^{-23}$

temperature T   = $TNOM + 273.15$

5 Manually enter the NF value in the Parameters table.

Manually Extracting BF

1 Using the ibicvsvb plot, pick a value of Vb in the linear portion of the Ib and Ic curves where the curves are as close as possible to parallel. The following figure shows an example.
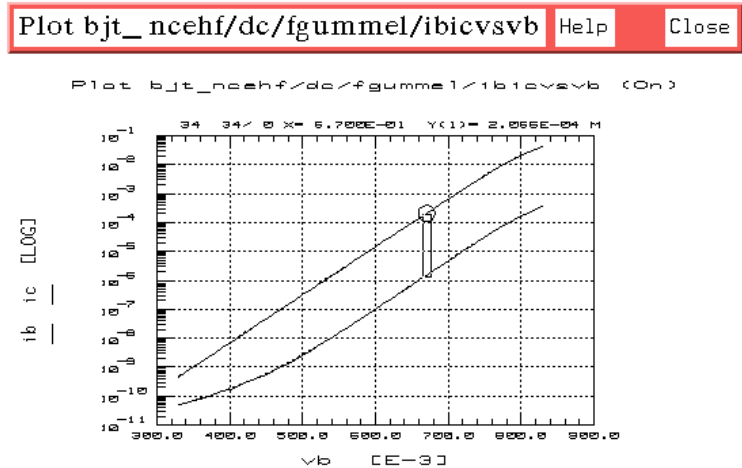
Figure 65   Manual Extraction of BF

$BF = \dfrac{Ic}{Ib}$ where both Ic and Ib are measured at the same Vb.

2   To determine the value of Ic at the chosen value of Vb,
    position the pointer on the trace at that value, and click
    middle to get a readout of the value at that point.

3   Repeat for Ib.

4   Subtract the logarithmic value of Ib from the logarithmic
    value of Ic to get the value of BF.

5   Manually enter the value of BF in the Parameters table.

Manually Extracting ISE and NE

1   Using the ibicvsvb plot, choose a linear portion at the low
    end of the Ib vs Vb curve, where the slight bending upward of
    the trace indicates the recombination in the base-emitter
    depletion region.

2   Use the Draw Diag Line feature to extrapolate the linear
    portion to the boundaries of the plot (the following figure
    shows an example).
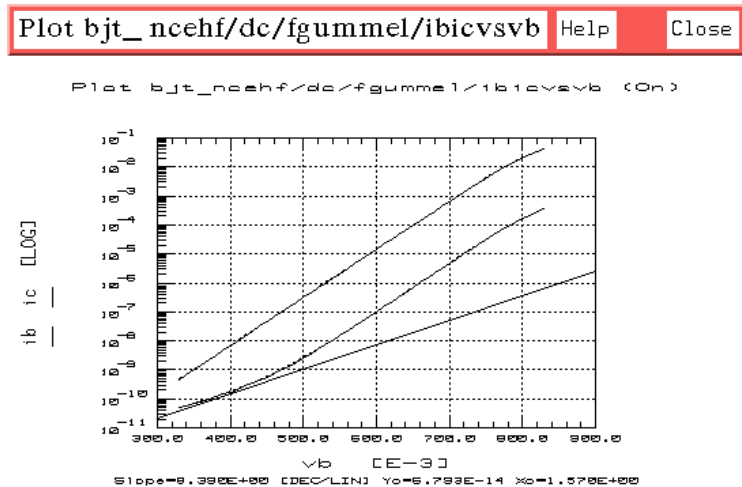
Figure 66   Manual Extraction of ISE Using Draw Diag Line

3   Read the Yo value from the annotations below the plot, and
    manually enter it in the Parameters table as the value for
    ISE.

4 Use a calculator to solve the following equation for NE. Note that this equation is only valid at low Vbe values where recombination takes place in the base-emitter space charge region.

$$NE = \frac{1}{V_T}\left[\frac{d \cdot VBE}{d \cdot (log_e)(I_B)}\right]$$

where: $V_T = \dfrac{kT}{q}$

and:

elementary charge q   = $1.60218 \cdot 10^{-19}$

Planck's constant k   = $1.38066 \cdot 10^{-23}$

temperature T   =   TNOM + 273.15

5 Manually enter the NE value in the Parameters table.

Manually Extracting IKF

A hardcopy plot is used for this procedure.

1 Choose **File > Print** from the ibicvsvb plot, to obtain a hardcopy of the plot.

2 Manually draw a diagonal line along the linear portion of the Ic curve in the medium range of Vbe. Extend the line beyond the trace at the top end, as illustrated.

3 Draw another diagonal line along the upper range of Ic where the high-level effects begin to dominate. Extend the low end of the diagonal line so that it intersects the first diagonal line, as in the following figure.

4 IKF is the value of IC at the intersection of the two diagonal lines. Put the mouse pointer on the trace at this point, and click middle to get a readout of the value.

5 Manually enter the value of IKF in the Parameters table.
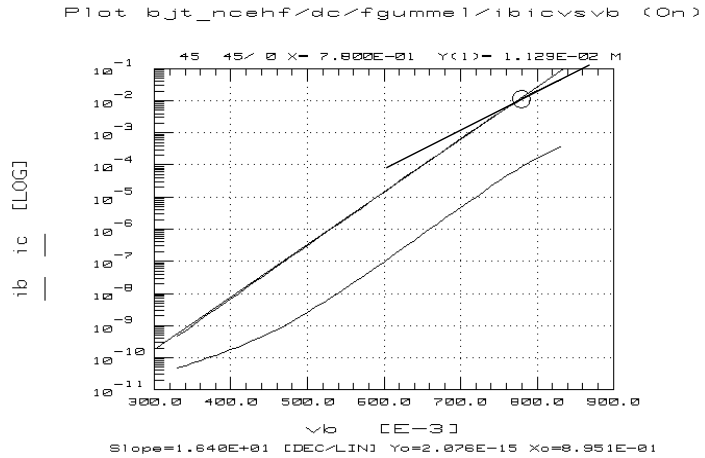
Figure 67   Manually Extracting IKF

Verifying the Manual Extractions

1 Select **Measure/Simulate > Simulate**.

2 Observe the ibicvsvb plot. The simulated and measured data should now be very close.

3 Now go back to "Optimizing IS and NF" on page 166. Perform that and the subsequent three procedures for optimizing BF, ISE and NE, and IKF.

4 Close the ibicvsvb plot.

Extracting NR, BR, ISC, NC, and IKR

The procedure for extracting these reverse parameters corresponds closely to the last procedure for extracting the forward parameters. The main differences are that IS is not extracted in the reverse direction, and that the rgummel data is used in place of the fgummel. As in the forward gummel extractions, this procedure uses several different transforms that isolate and optimize the individual parameters in the regions where they are most dominant. Note that extractions based on the rgummel measurement make the assumption that Ie = −(Ic + Ib).

Table 20   Reverse parameters extracted using the rgummel setup

Parameter	Description
NR	Reverse current emission coefficient (models the deviation of the collector-base diode junction from the ideal).
BR	Ideal maximum reverse beta.
ISC	Base-collector leakage saturation current (the value of the base current at zero base-collector voltage).
NC	Base-collector leakage emission coefficient (models the recombination of the base current in the base-collector space charge region at low base-collector voltages).
IKR	Reverse knee current (models the drop in reverse beta at high emitter currents due to high-level injection).

Initial Extraction

1 Select the **rgummel** setup.

2 Select **Extract/Optimize > nrextract > Execute** to extract the initial value for NR.

3 Select **rgextract > Execute** to extract initial values for BR, ISC, NC, and IKR.

4 Select **Plots > ibievsvc > Display Plot**.

5 Select **Measure/Simulate > Simulate**.

6 Observe the ibievsvc plot. The simulated data should be reasonably close to the measured data. If so, proceed to "Optimizing NR."

7 If the simulated data does not match fairly well with the measured data, the optimizer may not converge properly. In this case, extract the parameters *manually* with the Draw Diag Line feature, as explained for the forward gummel parameters under "Manual Extractions (Optional)" on page 169. Then come back to the optimization procedures given below.

Optimizing NR

This optimization is performed over a section of the Nr curve at medium Vce values, avoiding the low and high current effects.

1 Select **Plots > nr_vs_vc > Display Plot**.

2 Choose a linear section of the curve at medium Vce values, as shown in the following figure. Avoid the low and high current effects.
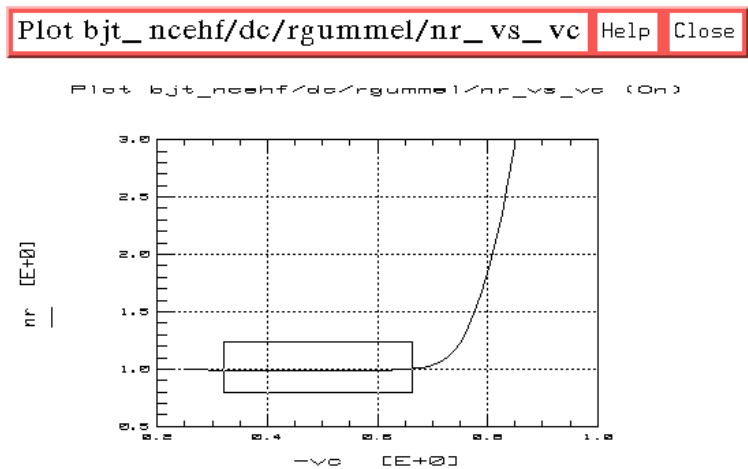
Figure 68   Appropriate Region of Trace for Optimization of NR

3 Use the X-low/X-high function to define the section of the curve to be optimized:

- Position the mouse pointer at the low end of your chosen range and click left, then at the high end and click left.

- From the plot menu select **Options > Copy to Variables**, to change the X-low/X-high values to the trace limits you selected.

4 Select **Extract/Optimize > optie_nr > Execute**. Observe the ibievsvc plot (Figure 69) to see the simulated Ib and Ie data converge to the measured data. The transform optimizes the value for NR using the section of the trace you defined, and enters it in the Parameters table.
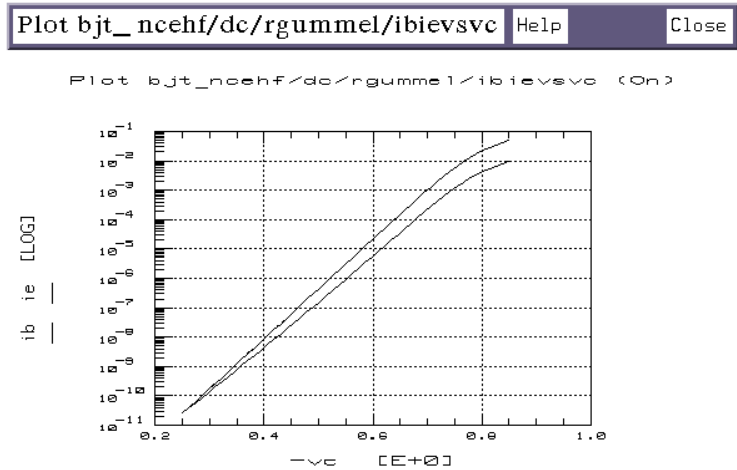
Figure 69   Measured and Simulated Ib and Ie Data, Well Converged

Optimizing BR

1 Use the same linear section of the trace as in "Optimizing NR" on page 176.

2 Select **opt_beta > Execute**. The transform optimizes the value for BR using the defined section of the trace, and enters the value in the Parameters table.

Optimizing ISC and NC

This optimization is performed at the low end of the curve.

1 Using the nr_vs_vc plot, choose a section at the low end of the curve, as in the following figure. If possible, look for a

transitional area where the slope begins to change, indicating the recombination in the base-collector space charge region.
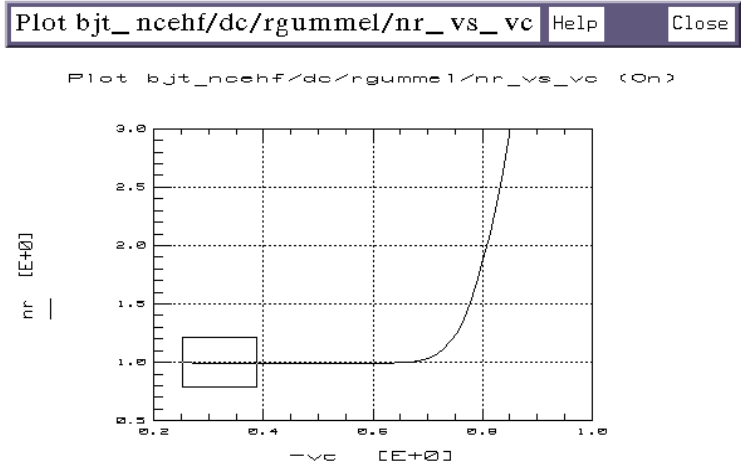
Figure 70   Appropriate Region of Curve for Optimization of ISC and NC

2 Use the X-low/X-high function to define the section of the curve to be optimized.

3 Select **optib_isc_nc > Execute**. The transform converges the simulated data to the measured, and enters the optimized values for ISC and NC in the Parameters table.

Optimizing IKR

This optimization is performed over the upper portion of the Nr curve.

1 Using the nr_vs_vc plot, choose a section at the high end of the measured curve starting with the transitional area where the slope begins to change. Refer to Figure 71: the portion of the trace in the box is the appropriate region for

optimization; the upper limit is set by the maximum value of Vc.
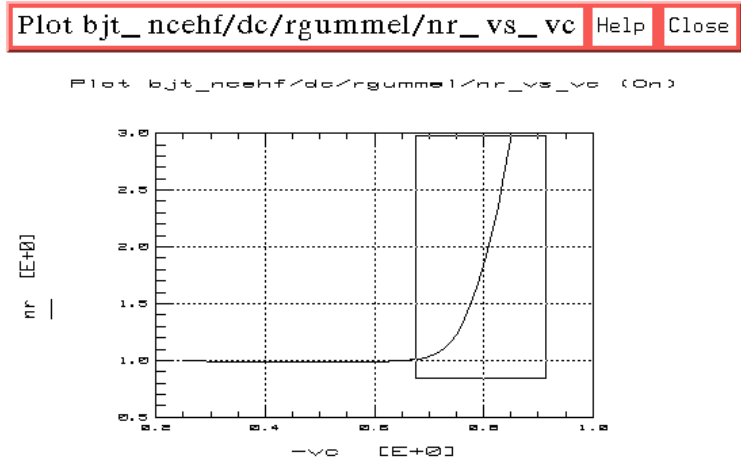
Figure 71   Appropriate Region of Trace for Optimization of IKR

2 Use the X-low/X-high function to define the section of the trace to be optimized.

3 Select **optie_ikr > Execute**. The transform converges the simulated data to the measured, and enters the optimized value for IKR in the Parameters table.

4 Close the plots.

Verifying the Predicted DC Performance

All the DC parameters have now been extracted. Before you proceed to the AC parameter extractions, it is important to verify that the extracted parameters do indeed predict the DC performance of the device. The AC performance of the device, and hence the AC extractions, are strongly dependent on the accuracy of the DC parameters.

Comparing the Measured and Resimulated fearly Data

1 Select the **fearly** setup.

2 Select **Plots > icvsvc > Display Plot**.

3 Select **Measure/Simulate > Simulate**.

4 Observe the plot. The resimulated data using the new parameters should be almost indistinguishable from the measured data.

5 Close the plot.

Comparing the Measured and Resimulated rearly Data

1 Select the **rearly** setup.

2 Select **Plots > ievsve > Display Plot**.

3 Select **Measure/Simulate > Simulate**.

4 Observe the plot. Due to the non-reciprocal nature of the base-emitter and base-collector junctions, the current levels of the measured and simulated rearly data will not coincide. SPICE assumes that the saturation current IS is identical for both junctions: since this is not in reality the case, it is seldom possible to have both the fearly and rearly data match simultaneously. However, the slope of the Ic vs Vce curve beyond saturation is independent of IS, and therefore the measured and simulated slopes should match in this region.

5 Close the plot.

Comparing the Measured and Resimulated fgummel Data

1 Select the **fgummel** setup.

2 Select **Plots > ibicvsvb > Display Plot**.

3 Select **Measure/Simulate > Simulate**.

4 Observe the plot. Even though the equations describing the forward gummel data are dominated by the forward gummel parameters, the reverse parameters do also have an effect. Therefore check that the forward gummel data resimulated using the new parameters still closely matches the measured data.

5 Close the plot.

Modifying the DC Parameters

You may obtain unexpected results from your comparisons between measured DC data and data resimulated using the new parameters. In this case, you might choose to make minor changes in the values of one or more parameters, to more closely match the simulated data to the measured data. If you do this, it is important that you check *all* the DC simulations, and modify other parameters if necessary, to compensate. This cycle must be continued until all simulations are acceptable, and you will probably have to make some trade-offs in the process. You will need to exercise judgment based on your knowledge of device physics and the particular device under test.

AC Extractions

The AC performance of the device, and hence the AC extractions, are strongly dependent on the accuracy of the DC parameters. It is essential that the DC parameters and the parasitic capacitance and resistance parameters are known to be good, before the AC extractions are done. If you have any doubts about the validity of these parameters, continue to refine the DC extractions before proceeding with the AC extractions.

Setting Initial Values for FC and XCJC

The final values for FC and XCJC will be extracted after the other AC parameters have been determined. Because FC and XCJC have an effect on the overall AC performance of the device, initial values are assigned to them before extraction of the other parameters.

- If you know the values of FC and XCJC for your device, enter those values into the Parameters table.
- If you do not know the values of FC and XCJC for your device, enter the value of FC as 0.5, and the value of XCJC as 0.7.

Extracting High-Frequency Transit Time Parameters

This procedure uses several different transforms to optimize the individual parameters in the regions where they have the most dominant effect. The parameters are extracted using both the h21vsvbe and ftic setups.

Table 21 Parameters extracted using the h21vsvbe and ftic setups

Parameter	Description
TF	Ideal forward transit time (models the finite bandwidth of the device in forward mode).
XTF	Bias-dependence coefficient of TF (models the minimum value of TF at low Vce and high Ic).
VTF	Voltage describing the dependence of TF on Vbc (models the effects of Vbc on TF).
ITF	High-current parameter effect on TF (models the decline of TF with high Ic).

Initial Extraction

1 Select the **h21vsvbe** setup.

2 Select **Measure/Simulate > Simulate**.

3 Select the **ftic** setup.

4 Select **Plots > ft_ic > Display Plot**.

5 Select **Extract/Optimize > Ft > Execute** to draw the Ft vs Ic data on the plot.

6 Select **AC_decoupled > Execute**. The transform calculates the values for TF, XTF, VTF, and ITF, and enters them in the Parameters table.

7 If the simulated curves do not converge closely to the measured curves, you can perform manual optimization procedures using the h21vsvbe setup, starting on the next page. The procedures make iterative changes to the values in the Parameters table to obtain a best fit of the simulated curves to the measured curves.

8 Close the ft_ic plot.

Optimizing TF and VTF

This is an iterative process that optimizes the convergence of the simulated data at the peaks of the curves. TF sets the height of both peak values of Ft (the peaks of both Vce curves). VTF sets the separation between the two curves at their peak Ft value.

1 Select **Model Parameters**. In the Parameters table, modify the values of both TF and VTF. These are "first guess" value changes that will be modified iteratively until you achieve acceptable results.

2 Select **DUT/Setups > h21vsvbe**.

3 Select **Plots > h21_vbe > Display Plot**.

4 Select **Measure/Simulate > Simulate**.

5 Observe the plot. The simulated curves should match the measured curves reasonably well at the peaks of both Vce curves, as in the following figure.
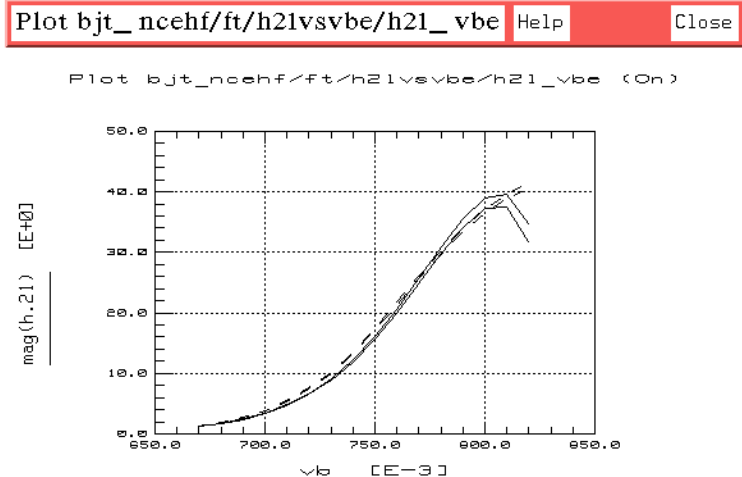
Figure 72   Measured and Simulated Vce Curves (Ft vs Ic)

6 If not, repeat the above steps as necessary to modify the TF and VTF values and resimulate, until the simulated and

measured curves at both values of Vce are close at the point where Ft is maximum.

7 Do not close the plot.

Optimizing ITF

This is an iterative process that optimizes the simulated data for the high end of the Ic curves. ITF predominantly affects the high Ic roll-off of the lower Vce curve, and also has an appreciable effect on the higher Vce curve.

1 In the Parameters table, modify the value of ITF. This is a "first-guess" value change.

2 Select the **h21vsvbe** setup.

3 Select **Measure/Simulate > Simulate**.

4 Observe the ft_ic plot from the ftic setup. The simulated curves should match the measured curves reasonably well in the region of high Ic roll-off.

5 If not, repeat as necessary to improve the convergence of the simulated data to the measured data in the region of high Ic roll-off, especially in the lower curve.

Optimizing XTF

This optimizes the convergence of the simulated data for the upper Vce curve. XTF value changes should be minimal.

1 After ITF is set, *sparingly* modify the value of XTF in the Parameters table.

2 In the h21vsvbe setup select **Measure/Simulate > Simulate**.

3 Observe the ft_ic plot. The upper simulated Vce curve should converge to the upper measured curve.

4 If not, repeat as necessary until the curves converge.

5 Close the ft_ic plot.

Extracting PTF

This procedure uses the H21 magnitude data measured in the sparm_bias setup to determine the value of PTF, the excess phase of TF (which is also the value of excess phase at F_t).

1 Select the **sparm_bias** setup.

2 Select **Plots > db_h21 > Display Plot**.

3 In the db_h21 plot, find the highest frequency where the H21 magnitude is 0 dB (see the following figure). If necessary, use the Draw Diag Line feature to extend the highest Vbe curve through the 0 dB line.
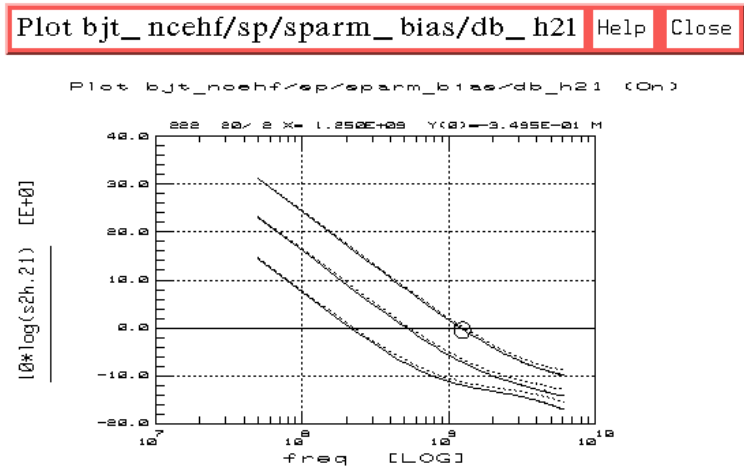
Figure 73   Determining F_t. Example Plot of H21 Magnitude

4 Position the pointer on the trace at this point and click middle to get a readout of the values. The frequency (x-axis value) at that point is F_t.

5 Select the **ph_h21** plot, and **Display Plot**. Figure 74 illustrates a typical H21 phase plot.

6 Find the location of F_t on the ph_h21 plot. Put the mouse pointer on the trace at this point and click middle to get a readout of the phase value.

Solve for PTF by subtracting 90° from the phase of H21 at F_t:

$$PTF\ (degrees) = |phase\ of\ H21\ @\ Ft| -90$$

7 Manually enter the calculated value of PTF in the Parameters table.
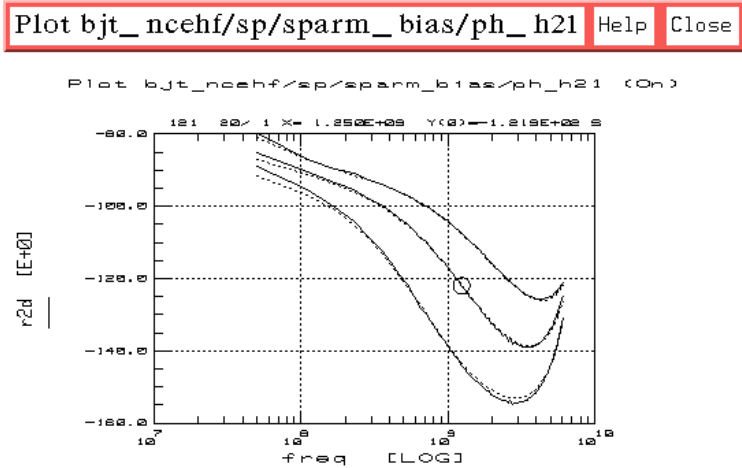
Figure 74 Typical Plot of H21 Phase, with Location of Ft

8 Close the plots.

Extracting IRB and RBM

This procedure uses the S11 data measured in the sparm_bias setup to extract the following parameters:

Table 22   Parameters extracted using the sparm_bias setup

Parameter	Description
IRB	Current where the base resistance falls halfway to its minimum value from its zero-bias value RB.
RBM	Minimum base resistance at high currents (models the effects of current crowding on the base resistance).

1 In the sparm_bias setup, select the **s11** plot, and **Display Plot**.

2 Select **Measure/Simulate > Simulate**. Observe the high-bias S11 curve on the plot. If the measured and simulated values agree at the resonant frequency (where the S11 curve crosses the real axis), no adjustment of RBM is needed. If the high-bias S11 curve does not cross the real axis, visually extrapolate to see where it would cross the real axis if it were extended: if measured and simulated values agree no adjustment of RBM is needed.

3 If the high-bias measured and simulated S11 values do not agree at the resonant frequency (or the extrapolated resonant frequency), modify the value of RBM in the Parameters table. This is a "first guess" value change.

4 Select **Measure/Simulate > Simulate**.

5 Continue to modify the value of RBM and resimulate until the measured and simulated values of S11 do agree at the resonant frequency.

6 Observe the medium-bias S11 curve on the plot. If the measured and simulated values agree at the resonant frequency, no adjustment of IRB is needed.

7 If the medium-bias measured and simulated S11 values do not agree at the resonant frequency, modify the value of IRB in the Parameters table and resimulate.

8 Continue to modify the value of IRB and resimulate until the measured and simulated values do agree at the resonant frequency.

9 The simulated low-bias S11 curve depends mainly on the value of RB extracted earlier, and should therefore already agree with the measured value.

10 Close the s11 plot.

Extracting CC

This procedure uses the S22 data measured in the sparm_bias setup to extract CC, the parasitic capacitance at the output of the device. This represents the capacitance between ground and the pad on which the device is epoxied, as well as any other stray capacitance to ground. This value is very difficult if not impossible to measure directly, therefore its value is determined indirectly using the following method.

1 In the sparm_bias setup select the **s22** plot, and **Display Plot**.

2 If the maximum frequency points (end points) of the measured and simulated S22 curves are comparable in value, no adjustment of CC is needed.

3 If the maximum frequency points of the measured and simulated S22 curves are not comparable in value, modify the value of CC in the Parameters table. This is a "first-guess" value change.

4 Select **Measure/Simulate > Simulate**.

5 Continue to modify the value of CC and resimulate until the maximum frequency points of the measured and simulated curves are comparable in value.

Extracting FC

This procedure uses the data measured in the ftic setup to extract FC, the coefficient for forward bias capacitance. The S11 data measured in the sparm_bias setup is also used to verify the value of FC.

FC models the continuity between the forward and reverse bias base-emitter capacitance. Therefore it has the most effect at the low end of the Vbe curves, in the region dominated by junction capacitances where the base-emitter junction capacitance is comparable to the forward diffusion capacitance.

1 Select the **ftic** setup.

2 Select **Plots > ft_ic > Display Plot**.

3 The simulated data should match the measured data reasonably well at the low end of the curves, in the region indicated in the following figure.
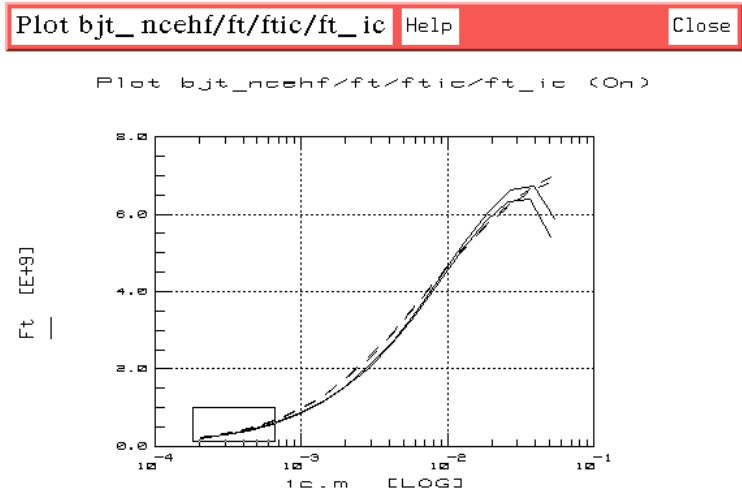
Figure 75   Region of Curves Where Junction Capacitances Dominate

4 If the simulated data does not match the measured data in the indicated region, modify the value of FC in the Parameters table.

5 Select **Measure/Simulate > Simulate**.

6 Continue to modify the value of FC and resimulate until the simulated data converges to the measured data in the indicated region.

7 Select the **sparm_bias** setup.

8 Select **Plots > s11 > Display Plot**.

9 Check the convergence of the simulated to measured S11 curves in the low-bias region. If they are well converged, this verifies that the extracted value of FC is good.

10 Leave the s11 plot open.

Extracting XCJC

This procedure uses the S11 and S22 data measured in the sparm_bias setup to extract XCJC, the fraction of base-collector capacitance connected to the internal base node. XCJC models the distributed effects of the base-collector junction. The effects are typically only noticeable at high frequencies.

1 In the sparm_bias setup select the **s22** plot, and **Display Plot**.

2 The simulated data should match the measured data at the higher frequencies.

3 If not, modify the value of XCJC in the Parameters table.

4 Select **Measure/Simulate > Simulate**.

5 Repeat as necessary to improve the convergence of the simulated to measured S11 and S22 curves at the higher frequencies.

Final Parameter Adjustments

All the BJT parameters have now been extracted. To verify the accuracy of the AC parameters, now resimulate all the AC setups using the final parameters. It may be necessary to make minor modifications to one or more of the parameters to achieve a better fit between the simulated and measured data. This is a somewhat nebulous task: the following instructions are guidelines that try to provide some structure. None of the parameters should require more that slight adjustment: if any of them do, some of the parameter values may be suspect.

Recall that the DC, capacitance, and parasitic resistance parameters have a significant influence on the simulated AC performance of the device. Therefore, if it is difficult or impossible to find AC parameters that adequately predict the AC performance, it may be necessary to modify some or all of these parameters. Note that, because of the inherent interaction between parameters, any change to one usually requires the modification of many. If too many parameters need to be modified by too much, it may be advisable to start the extraction process over.

LE, LB and LVIA

Use the S-parameter plots in the sparm_bias setup, starting with s11. Cautiously adjust the values of these parameters in the Parameters table and resimulate, until the simulated S-parameter curves match the measured ones. The values of the parasitic inductance parameters vary the length of the curves, particularly the S11 curves. This is because of the BF+1 factor in the impedance transformation from emitter to base, which causes the inductances at the emitter to have much more of an effect on the input impedance, and consequently on S11.

RC

Use the s22 plot in the sparm_bias setup. Cautiously adjust the value of RC in the Parameters table and resimulate, until the simulated S22 curves match the measured ones. The effect of RC is most prominent at the high-frequency end of the S22 curves. Varying RC affects the loss of S22, therefore the value of RC can be adjusted until the high-frequency ends of the simulated and measured S22 curves line up.

TF

Use the h21_vbe plot in the h21vsvbe setup. Adjust the value of TF in the Parameters table and resimulate, to improve the match between the peaks of the simulated and measured curves.

XCJC and FC

The requirements for extracting these parameters are less rigid than for other more explicitly defined parameters. Therefore, you can modify either or both of these values in the Parameters table to adjust the overall simulated AC performance if necessary.

RE

Use the s11 plot in the sparm_bias setup. Although RE has been determined through the DC extraction using the reflyback data, you may need to adjust its value slightly to accommodate the AC

performance. You can do this by monitoring the simulated loss in the S11 curves. Note that in a fashion identical to LE the effect of RE on the input impedance is increased by the factor BF+1. This results in S11 being very sensitive to RE, especially at bias values where BF is large. Be very careful when you adjust this parameter!

IS and NF

Be particularly careful here. These two parameters have a *dramatic* effect on the overall performance of the device. If, during the course of fine-tuning the final parameter values, you decide to adjust either IS or NF, you will need to perform the extraction process again, based on the new values for IS and NF.

Storing the Model

This procedure stores the model in IC-CAP, including all the device-specific measured and simulated data. Once you have developed the device model, it can be used as a component in the Agilent RF and Microwave Design System (MDS). You can use it for circuit simulation, inserting it into circuits under design to emulate the characteristics of your device.

If you will be modeling large numbers of devices, it is worthwhile to organize your directory structure according to device type, geometry, and batch or wafer.

1 Use the directory you created at the beginning of the procedure, when you first opened the *bjt_ncehf.mdl* file.

2 From the bjt_ncehf:1 main menu select **File > Save As**.

3 A dialog box is displayed, as illustrated.
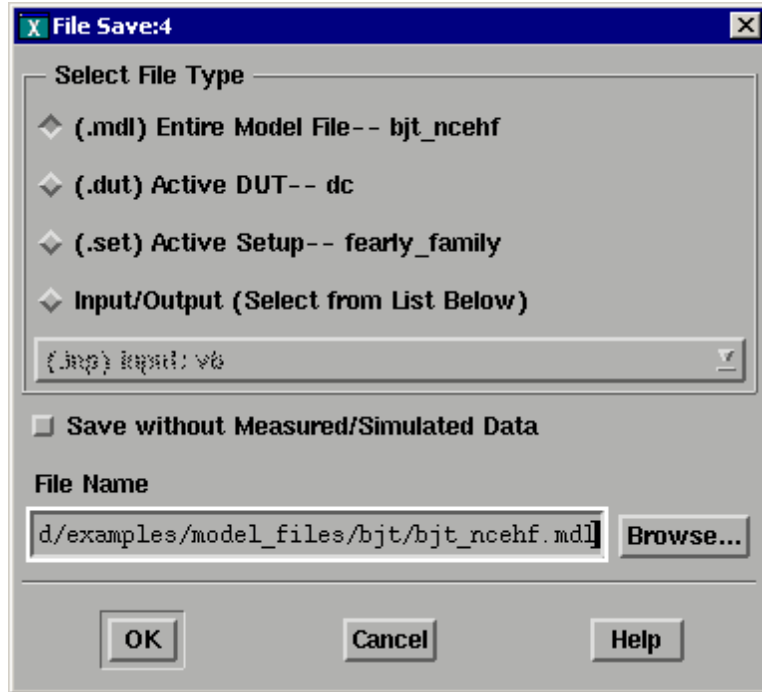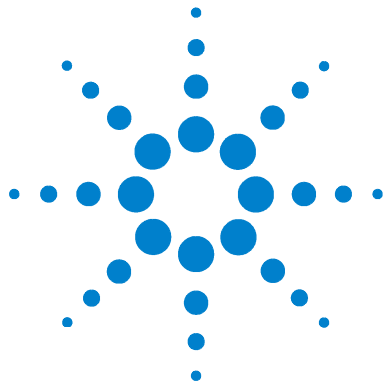
Figure 76   Save As Dialog Box

4 Select the File Type (*.mdl* for a complete model) by clicking the corresponding radio button.

5 Select a new filename that associates this model with the device, for example *bjt3.mdl.* Type the full directory pathname and the filename into the dialog box. Select **OK** to store the data and close the dialog box.

4

Agilent Root MOSFET Model Generator

This chapter documents the Agilent 85194B Agilent Root MOSFET model generator software. It provides an example procedure that takes you step-by-step through a complete data acquisition and model generation.

The complete procedure includes all the instructions needed to generate a new device model by modifying variable values in macros and factory default tables. After you have set these variables, subsequent measurement and model generation procedures are much faster.

Model Description

The Agilent Root MOSFET model applies to MOS devices the same database modeling principles as those used in the Root FET model. This three-terminal model can be applied to vertical, LD, and power MOS devices, where substrate effects do not need to be a part of the model. It can be used for both NMOS and PMOS devices. As with the Root FET model, the Root MOS model is generated from small-signal S-parameters measured at numerous bias points over the entire operating current-voltage (IV) plane.

Based on current and power dissipation device compliances you provide in software, the data acquisition system calculates the safe operating range for the device. Within this region, the model takes data adaptively at multiple bias points, depending on the specific nonlinearities of the device. Measurements are densely spaced in the most nonlinear regions, such as at the knees of the IV curves and the onset of breakdown, and less densely spaced in linear regions.

The data set of internodal nonlinear current and charge components at each bias point is then mathematically generated and stored. The model is thus generated without the usual need for simulation and optimization as required by all previous empirical models. The Root MOS model may thus capture nonlinear device behavior for any MOSFET device for which measured data can be taken but for which there are no good physical or empirical models.

The data acquisition system sets the DC bias levels and controls the system hardware to take the DC and S-parameter measurements, based on your inputs. The model generator processes the measured data and generates the lookup tables required by a circuit simulator. It produces data files that are directly readable by the Agilent RF and Microwave Design System (MDS). Both IC-CAP and MDS can be run simultaneously in the X Windows environment. During a simulation, the tabular state-function data from the generated Agilent Root MOS model is interpolated using multidimensional spline functions to emulate the terminal characteristics of the device.

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP.

Getting Started in IC-CAP

This example procedure assumes the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"
  - Wiltron 360-based system—Refer to your Wiltron documentation
- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.
- The IC-CAP software has been properly installed on the computer.
- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename the SMUs to correspond with the MOSFET terminals

Rename a medium-power SMU such as MPSMU1 as VG (gate supply)

Rename a high-power SMU such as HPSMU3 as VD (drain supply)

Rename the GNDU unit (which must be connected to the device source) as GND

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on opening the IC-CAP software and accessing the model file you need. The default path name and filename for the Agilent Root MOS model file used as a starting point in these procedures is:

/examples/model_files/mosfet/HPRootMos.mdl

Then return to this chapter and continue with the procedure.

NOTE	Do not install the device yet. The procedure will instruct you to install it after extracting the measurement port resistances.

Before you continue with the IC-CAP procedure, make a new directory in an appropriate location in the UNIX structure, using the mkdir command, to store the data and model files that will be generated. Give the directory a name that will associate it with the device you are going to model. Use the cd command to change to the new directory in UNIX.

MNS Link

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP. The link is set up for MDS version B.07.00, and will need to be modified whenever MDS is updated. For the link to work, the path and environment variables need to be set correctly. This is done at the factory for the path and environment of a typical MDS B.07.00 system. Only minor changes to reflect unusual MDS paths or updated MDS versions are likely to be required. If you do need to modify your UNIX files to reset the MNS link, follow these steps to set the root MDS directory on your system and add *$MDSROOT/mdsbin* to the path:

For shell or kshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory to automatically set the environment variables:

MDSROOT=/mds_b.07.00 *(depending on the version and location of MDS)*
export MDSROOT
PATH=$MDSROOT/mdsbin: "$PATH"
export PATH

2 Also add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the MNS simulator:

mns mns /mds_b.07.00/hp 85150/lib/mns800 ""
CANNOT_PIPE

For cshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory:

#Setup for mns simulator:

setenv MDSROOT /mds_b.07.00
setenv PATH $MDSROOT/mdsbin: "$PATH"

2 Add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the MNS simulator:

mns mns /mds_b.07.00/hp 85150/lib/mns800 ""
CANNOT_PIPE

Using the Agilent Root MOS Model

This chapter provides an example procedure that takes you step-by-step through a measurement and model generation of an NMOS device. Values you use in generating your own device model will depend on the particular characteristics of your device: use its data sheet (if available) as a reference.

This example is provided as a starting point. It is a straightforward procedure that uses only a subset of IC-CAP's total capabilities. The complete procedure documented gives all the steps necessary to configure IC-CAP with the parameters for preverification measurements, parasitic extractions, and the main data acquisition. Once a model has been developed for a particular device type, the built-in scaling feature allows you to apply it to any device with the same process and a different total gate width and/or number of fingers. The modeling procedure then becomes more simple, and can be accomplished in a few easy steps.

NOTE	Although you will modify the variable values in factory default files during this procedure, do not under any circumstances change the *names* of any variables or setups. These names are used by the software in the data acquisition and model generation process.

NOTE	The example procedure measures the MOSFET under test in a common-source configuration.

The example procedure gives detailed instructions on how to:

- Initialize device parameters.
- Measure and extract the system measurement port series resistances.
- Make a DC measurement to preverify device DC performance.
- Calibrate the network analyzer for both swept and CW measurements.

- Make an S-parameter measurement to preverify device performance at high frequencies.

- Measure and extract the device parasitic resistances.

- Set up and initiate the main data acquisition. Plot the measured data.

- Set up and initiate the model generation. Plot the calculated state functions.

- Verify the model against measured data.

- Store the data and model files for circuit simulation in Agilent MDS.

<table>
<tr><td>NOTE</td><td>Measurement parameters and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct results.</td></tr>
</table>

The procedure provides guidelines on setting values to measure your device. However, the values you use will be based on your knowledge of your device and on information in its data sheet, as well as on the guidelines presented here: they may or may not correspond with the example values.

The illustrations of plotted data are provided as visual examples of possible results. They are not intended to represent a single device, nor do they necessarily correspond with the values given in the procedure. The plots can be used to check the reasonableness of your own data, which should appear similar in shape but not necessarily coincident in values.

The DUT and Setups

The Model window is the central access point for all the procedures in the measurement and model generation process. The Model window includes the DUT/Setup panel and the file tabs to access the Model Parameters table, the model variable table, and other needed IC-CAP features.

The illustration shows the DUT/Setup panel for the model generator file *HPRootMos.mdl*. The DUTs (for example *pre_verify*) indicated by the inverted triangles are groupings of

similar measurement setups used to make related measurements or extract related model parameters. The setups (for example *idvd_vg*) contain the information used to define the inputs and outputs for each measurement and extraction, as well as the measurement hardware configurations and the transforms and plots associated with each setup.
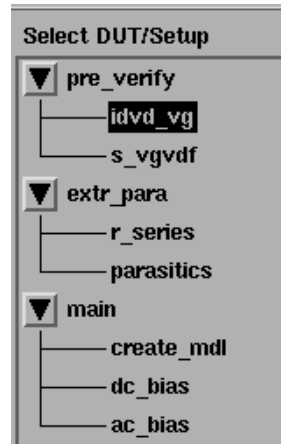
Figure 77   Agilent Root MOSFET DUT/Setup Panel

Initializing Device Parameters

You begin the procedure by giving the device an identifying name, and entering the number of fingers and gate width. For this you use the init_parameters macro. When this macro is executed, it also initializes internal program functions that are essential in the model generation process, and it provides annotations for the data plots. (IC-CAP macros are used to combine or simplify operations.)

1 Click the **Macros** tab to display the macro names.

2 From the Select Macro list, select **init_parameters**, and **Execute**.

3 In the dialog box that appears, enter a name to identify the device under test, then click **OK** or press Return.

4 In the next dialog box, enter the number of gate fingers.

5 Next enter the gate width in meters.

6 The next dialog box prompts you for a directory name. Click **OK** to store the model files in the directory shown, enter the name of another existing directory. Click **OK** or press Return.

7 The macro is executed, and the values you selected are entered in the model variable table. You can open the table by clicking on the **Model Variables** tab.

NOTE   Do not edit the init_parameters macro. The macro includes *executable code;* modifying it can cause problems in the program.

Setting the Extraction Frequency

You use the model variable table to set the frequency at which the model will be generated. To edit the variable table, move the mouse pointer to the appropriate variable value, click left, and move the pointer over the old information to highlight it. Then type in the new value over the old.

Name	Value
Freq	2.0G
Num_fingers	5
Width	3.5um
r1s	400.0m
r2s	400.0m
SIMULATOR	mns
ANNOTATE_PLOTS	YES
ANNOTATE_FILE	icplotnotes
ANNOTATE_AUTO	YES

Figure 78   Model Variable Table

1 Enter the CW frequency you intend to use for the data acquisition and model generation. This must be the same as the frequency of the network analyzer CW calibration you will perform before doing any S-parameter measurements.

Refer to the explanation under "Calibrating the Network Analyzer" on page 214, which explains how to perform a CW calibration and how to decide the CW frequency.

2 Make sure the Value listed for SIMULATOR in the model variable table is mns, to ensure that the mns link is in place to allow verification of the model in IC-CAP.

3 Do not change the other values in the table at this time.

Values in the Parameters Table

The illustration shows the Model Parameters table, with the initial values for the extrinsic parasitic capacitances and inductances. The table is presented here so that you can enter values for these extrinsic parasitics based on your knowledge of the device, or on estimates, or on measurements of the open fixture. In addition, you can set limits by entering minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values.

Param Name	Value	Min	Max
cpds.c	0.000		
cpgd.c	0.000		
cpgs.c	0.000		
ls_ext.l	0.000		
ld_ext.l	0.000		
lg_ext.l	0.000		

Figure 79   Model Parameters Table

To access the Parameters table, click on the Model Parameters tab.

To change a value in the Parameters table, move the mouse pointer over the current entry and click left to highlight it, then type in a new value.

The default for all the c values is 0.000. A reasonable capacitance value to enter for a pad on a wafer might be 50 fF.

The *l_ext.l* values are the bond inductances for the three terminals. The default value for all three is 0.000. If you know or can estimate the bond inductances for your device based on wire length or grid size, enter appropriate values.

Extracting the Measurement Port Series Resistances

This procedure need be performed *only* following initial installation of the system or if changes are made in the system hardware.

The procedure measures a short circuit at the normal device measurement interface plane. IC-CAP controls the DC source/monitor to force a gate current and monitor the gate voltage. The primary measurement is at port 1, and the port 2 measurement is synchronized to the port 1 measurement. You set the measurement parameters by defining the instrument state and the DC input values, as explained in the next pages.

The software calculates the series resistances from the outputs of the bias networks to the inputs of the device under test. The resistance values are then incorporated into the model file, and their effects are removed from the device response during the modeling process. The default port resistance for both port 1 and port 2 is 0Ω.

Note that the first part of the procedure differs for probe station and in-fixture measurements.

Setting Up for the Measurement

1 If you are using a probe station, position an impedance standard substrate (ISS) on the chuck. Place both the port 1 and port 2 probes on a short circuit standard.

2 If you are using a fixture, insert the short from the in-fixture calibration kit.

3 From the *DUT/Setup* panel select the r_series setup.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Set Integ Time to M (medium). Then return to this chapter and continue.

Setting the r_series Inputs

Use the following guidelines to set the ig and id inputs (most will not need to be changed from the default settings):

Table 23   For the ig input

Set	To
Mode	I
To Node	G (gate)
From Node	GROUND
Compliance	as high as 5 to 10V, because the device is not yet connected
Start, Stop	values that define a current range within the limits of the bias network
# of Points	a value sufficient to make several measurements across the current range
The Step Size will be set automatically	

- The Unit name must match the name of the SMU connected to the device gate (probably VG). This is also the SMU unit name set in the hardware window. Refer to the appropriate chapter as follows:

  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

- With a current input, compliance refers to voltage.

CAUTION   The bias networks can be destroyed if IC-CAP measurement current values are set too high! Be sure you know the maximum current ratings of your bias networks, and do not set current values beyond these limits. (The standard bias networks for the Agilent 85122A and 85123A parameter extraction test systems are rated at 0.5 amp.)

Table 24   For the id input

Set	To
Mode	I
To Node	D (drain)
From Node	GROUND
Unit	VD
Compliance	the same as for ig
Sweep Type	SYNC
Ratio	1.000, so that the id sweep is synchronized to the ig sweep
Offset	0.0
Master Sweep	ig, so that the id sweep is synchronized to the ig sweep

- The Outputs are factory set, and should not be changed.

Measuring and Extracting the Port Series Resistance Values

1 Under Measure/Simulate select **Measure**.

2 When the measurement is complete, the Measuring light on the DC source/monitor goes out and the IC-CAP wristwatch icon goes away.

3 Select the **Extract/Optimize** tab.

4 Select **compute_rseries > Execute**. The transform calculates the port series resistances from the measured data and enters the values into the model variable table as r1s and r2s.

Installing the Device

CAUTION   Ground yourself with an antistatic wrist strap to reduce the chance of electrostatic discharge. (Note that the Agilent 85122 or 85123 system work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.)

Install the device carefully on the probe chuck or in the fixture. Be sure to handle the device as little as possible, to avoid damaging it. Once the device is in place, avoid bumping the test station.

CAUTION   Remove any high-intensity light sources such as microscope light before taking a measurement. One simple method of blocking light is to place an opaque box on the fixture directly above the device.

Preverifying the Device DC Data

This procedure uses the idvd_vg setup to measure drain current and gate current with respect to drain voltage, at several values of gate voltage. The drain voltage is swept from 0V to the upper limit of its normal operating range. This is an opportunity to take a first look at the device performance and explore its characteristics. This also lets you select the most appropriate measurement range for the main data acquisition and other measurements.

In addition, this is a quick way to verify that the device DC characteristics are good, before starting a complete data acquisition and modeling generation. The data from this measurement can also be used for later comparison with data predicted by the generated model. It is not necessary to do the preverification measurement every time you perform a model generation, especially if you are using a data sheet.

Device Configuration: The input values you set in this procedure configure the device as shown in the following figure.
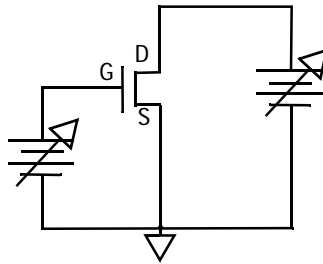
Figure 80   MOSFET Bias Configuration for DC Preverification

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the r_series measurement. Use the same settings.

Then return to this chapter and continue.

Setting the idvd_vg Inputs

This procedure defines the input signals to be applied to the device under test for the DC preverification measurement.

The actual settings you use will be dependent on the compliance limits for both the device and the SMUs/bias networks, and on the range of measurements you wish to take. Since this measurement is intended only to preverify the device, the values can be quite conservative and need not stress the device to the boundaries of its performance.

If you are measuring a series of devices of the same type and have already set the input parameters, go directly to . Otherwise, select the idvd_vg setup use the following settings as a guideline:

Table 25   For the vd input

Set	To
Mode	V
Compliance	no greater than the maximum breakdown current value for the device
Sweep Type	LIN, to provide a linear voltage sweep from start to stop voltage values
Start	0.0
Stop	at or below the upper limit of the device's normal operating range
# of Points	sufficient to clearly display the measurement results
The Step Size will be set automatically	

Table 26   For the vg input

Set	To
Mode	V
Compliance	no greater than the maximum allowable gate current value.
Sweep Type	LIN, to provide a linear voltage sweep from start to stop voltage values
Start, Stop, # of Points	such that three to five values (points) will be measured, in a range from approximately Vgs=Vthreshold to Vgs=3 or 4V. Set Start to a value close to Vthreshold, but not so close that the curve will be difficult to discern.
The Step Size will be set automatically	

NOTE	The vd and vg ranges apply to an NMOS device. Recognize that for a PMOS device both vd Start and vg Start will be negative, and Stop will be 0.000.

- The Unit names must conform to the names you set in the hardware window when you configured IC-CAP to recognize the system hardware (probably VD and VG). They must also correspond with the actual SMU (source/monitor unit) connections from the Agilent 4142. Since the device source terminal is grounded, only the drain and gate inputs need to be set.

- You provide compliance values to limit SMU output voltage or current and thus prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used. With a voltage input, compliance refers to current.

- The Outputs are factory set, and should not be changed.

Measuring and Plotting

This procedure triggers the DC source/monitor measurement and monitors the results.

NOTE	Before making the DC measurement, manually put the network analyzer in hold sweep mode by pressing STIMULUS > MENU > MORE > HOLD on an Agilent 8510, or MENU > TRIGGER MENU > HOLD on an Agilent 8753. Be sure to keep a UNIX window open so that you will see any compliance error that may be reported by the DC source/monitor.

1 From the Plots tab, select **Display All** (to the left of the setup window). Two plots are displayed.

2 Select the **Measure/Simulate** tab, then select **Measure**. When the measurement is complete, the Measuring light on the DC source/monitor goes out, the IC-CAP wristwatch icon disappears, and the measured data is displayed in the plot window.

3 Check the id_vd plot. The data displayed should resemble that shown in the following figure, with the lowest curve

close to the X-axis to show the threshold characteristics. If the data indicates the device is not operating in its normal range, go back to the inputs and change the gate voltage values. (If more than one curve is low, change vg Start to a higher value. Or if all the curves are high, change vg Start to a lower value.) Then select **Measure** again.
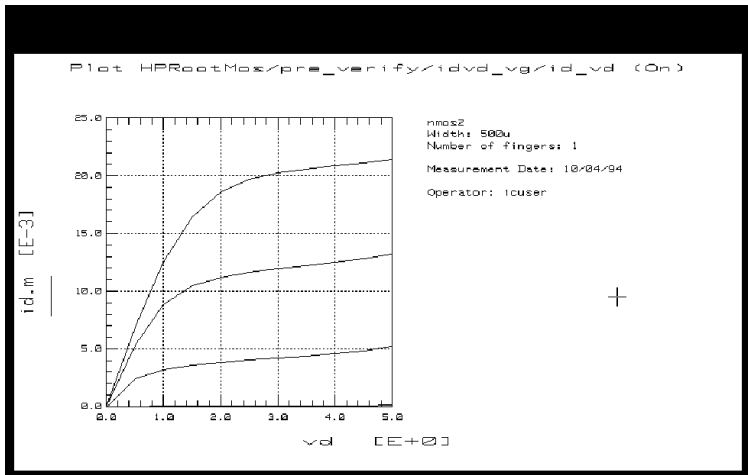
Figure 81   Example Measured pre_verify id_vd Data

4 If the measured data looks wrong for your device, check the probe contacts, the bias connections, and the measuring instruments. Make sure the device is oriented properly, with the right ports connected to the right terminals. If none of these resolves the problem, the device may be defective. For example, if all the curves are superimposed, the gate may be shorted.

5 To save the measured data for later comparison with data predicted by the extracted model, choose **File > Save As** and a filename with the suffix *.set* (for example *dc_test.set*). Then click **OK**.

6 The following figure illustrates a typical id_vd plot for a PMOS device.
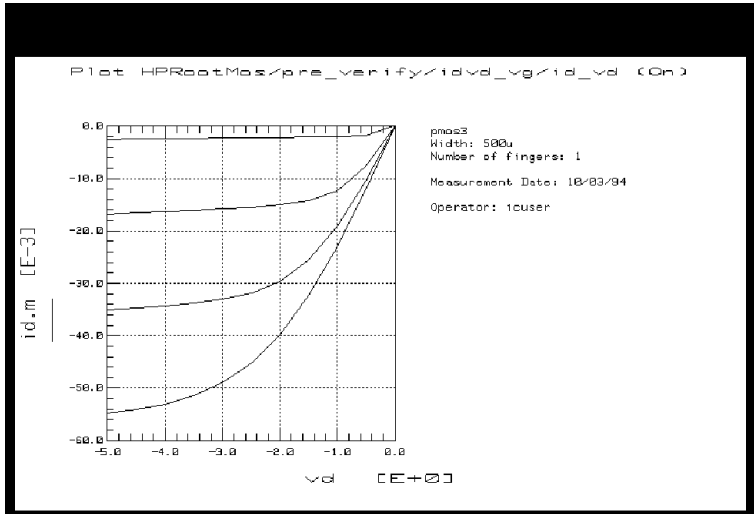
Figure 82   Example Measured id_vd Data for a PMOS Device

7 Close the plots.

Calibrating the Network Analyzer

It is important to calibrate the network analyzer before you perform any S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and model generation.

The Agilent Root MOS measurement procedures require two network analyzer calibrations: a swept broadband cal and a CW cal. The broadband cal is used for the S-parameter preverification and parasitics measurements. The CW cal is used for the main data acquisition.

For a system with an Agilent 8510 network analyzer, this procedure uses the frequency list cal mode for both the broadband and CW calibrations. The two calibrations can be done with only one set of standards measurements, by making the CW cal a subset of the broadband cal. For a system with an Agilent 8753 or 8720 network analyzer, two separate calibrations must be performed.

Frequency list calibration procedures for an Agilent 8510-based system and an Agilent 8753-based system are provided in the next few pages. A CW calibration subset procedure for an Agilent 8510-based system is also provided.

Calibration for an Agilent 8753-Based System

Although the Agilent 8753 has frequency list capability, IC-CAP does not support the subset calibration technique used in an Agilent 8510 system. Therefore it is necessary to perform two separate calibrations for an Agilent 8753- or 8720-based system: one frequency list calibration and one CW calibration. A separate set of standards measurements must be made for each calibration.

Frequency List Mode

Frequency list lets you define an arbitrary list of frequencies in one or more segments. During the measurement sweep the synthesizer is phase-locked at each frequency point, as in the stepped-sweep mode. The advantage of frequency list mode for use in a table-based model such as this is that it lets you define a smaller number of measurement points than for a step or ramp frequency cal. This reduces the total measurement time. A # of Points of 11 provides adequate measurement resolution as well as a quite fast total measurement time.

For the purpose of this model you will define two frequency segments in the list, a swept broadband segment and a CW segment. Set the broadband frequency list segment range at least as wide as the operating range of the device.

Set the CW frequency segment to the frequency you intend to use for the data acquisition, the same frequency you set in the model variable table at the beginning of this procedure. This frequency should be above 200 MHz to eliminate the effects of test set coupler roll-off at low frequencies, and low enough to eliminate bond inductances and other parasitic inductance considerations.

Based on the broadband cal, use a calibrated measurement point equal to the start frequency plus an integer multiple of the step size. An example frequency might be 600 MHz.

200 MHz (start freq) + 1 · 400 MHz (step size) = 600 MHz

Frequency List Calibration for an Agilent 8510-Based System

If you are using an Agilent 8510-based system, follow the steps below.

1 On the network analyzer press PRESET.

2 To define a linear frequency list, on the network analyzer press STIMULUS > MENU > MORE > EDIT LIST > ADD. A default frequency list segment is presented.

3 To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as M/μ) at the right of the keypad. Similarly, press STOP and set the stop frequency.

| NOTE | Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys in the STIMULUS area of the front panel. |

4 Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range. Keep in mind that the fewer the number of points, the less time the measurements take.

5 Press DONE to return to the frequency list menu. The segment you just defined is listed on the screen.

6 Press DONE to return to the main stimulus menu. Press FREQUENCY LIST to activate the frequency list mode.

7 Press PRIOR MENU > MORE > CONTINUAL to set a continual stimulus sweep.

8 Press PRIOR MENU > POWER MENU, and set the desired power level. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If the power level at the sampler goes above ≈−8 dBm, an **IF OVERLOAD** error message is displayed and you

will need to reduce the source output power. The default network analyzer power level is 0 dBm.

9 For a device with power drop-off at higher frequencies, you may wish to set a power slope using the stimulus menus. An appropriate power slope would be in the region of 2-3 dB/GHz.

10 Press RESPONSE MENU > AVERAGING ON/restart, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. Although the default is 256, you may wish to use an averaging factor as low as 16 to reduce overall measurement time.

11 If the cal kit constants for your calibration kit are not loaded into the network analyzer, load them from disc now by pressing DISC > LOAD and specifying the data type and file name.

12 If you wish to modify one of the internal cal kit definitions (see Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"), do so now.

13 Press CAL > CAL1 or CAL2 (depending on your cal kit), and perform a two-port calibration, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete.

14 At the end of the calibration sequence, the cal set numbers are listed in the softkey menu. Press CAL SET #1 to store the calibration in cal set #1. (Or use another available cal set.)

CW Calibration Subset for an Agilent 8510-Based System

1 To create a CW frequency subset of the broadband frequency list calibration, press CAL > CORRECTION ON > CAL SET 1 (this turns on the broadband cal you just completed), then MORE > MODIFY CAL SET > FREQUENCY SUBSET. Use the SUBSET: START and SUBSET: STOP softkeys in the SUBSET menu (*NOT* the front panel keys) to set both start and stop to the same frequency.

2 Press CREATE & SAVE, and store this calibration in cal set #2 or another available cal set.

3   Note the numbers of the cal sets where you save the calibrations: you will need to list one of these numbers as part of the instrument state for *each* of the network analyzer measurements.

NOTE	For more detailed information, refer to the Agilent 8510 *Operating and Programming Manual*.

Frequency List Calibration for an Agilent 8753-Based System

If you are using an Agilent 8753-based system, follow the steps below.

1   On the network analyzer, press LOCAL to gain front panel control. Press PRESET to return to a known standard state.

2   If you are using a system with the 6 GHz receiver option and you wish to measure in the 3 GHz to 6 GHz range, press SYSTEM > FREQ RANGE 3GHz6GHz.

3   To define a linear frequency list, on the network analyzer press MENU > SWEEP TYPE MENU > EDIT LIST > CLEAR LIST > YES > ADD. A default frequency list segment is presented.

4   To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as M/μ) at the right side of the keypad. Similarly, press STOP and set the stop frequency.

NOTE	Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys on the front panel.

5   Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range, then press the x1 terminator key. Keep in mind that the fewer the number of points, the less time the measurements take.

6   Press DONE to return to the edit list menu. The segment you just defined is listed on the screen.

7 Press DONE to return to the sweep type menu. Press LIST FREQ > SINGLE SEG SWEEP to activate the frequency list mode and set the analyzer to sweep the segment you defined.

8 Press RETURN > RETURN > TRIGGER MENU > CONTINUOUS to set a continuous stimulus sweep.

9 Press MENU > POWER and set the desired power level, then press the x1 terminator key. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If a receiver input is overloaded (>+14 dBm), the analyzer automatically reduces the output power of the source to −85 dBm and displays the error message **OVERLOAD ON INPUT (R, A, B) POWER REDUCED**. In addition, the annotation P↓ appears in the left margin of the display to indicate that the power trip function has been activated. When this occurs, reset the power to a lower level, then toggle the SOURCE/PWR on/off softkey to switch on the power again.

10 Press AVG > AVERAGING FACTOR, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. A good default averaging factor is 256. To speed your measurements, you may find it convenient to set an averaging factor as low as 16. Press AVERAGING ON.

11 You can further reduce the noise floor by reducing the receiver input bandwidth. Press IF BW in the average menu, and enter one of the following allowed values in Hz: 3000, 1000, 300, 100, 30, or 10. A tenfold reduction in IF bandwidth lowers the measurement noise floor by about 10 dB; however, the sweep time may be slower. For more information on averaging and the different trace noise reduction techniques, refer to the Agilent 8753 operating manual.

12 Press CAL > CAL KIT, and select the appropriate default or user-defined cal kit for your calibration devices.

13 If you wish to modify an internal calibration kit definition (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"), do so now.

14 Press CAL > CALIBRATE MENU, and perform the calibration of your choice at the analyzer front panel, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete. A full two-port cal provides the greatest accuracy. A TRL* or LRM* cal is an appropriate alternative for in-fixture measurements (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"). Omit isolation cal. Press DONE, and save the cal in the register number specified in the instrument options table.

15 Detailed procedures for measurements of calibration standards are provided in the Agilent 8753 operating manual.

CW Calibration for an Agilent 8753-Based System

Refer to Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" for a CW calibration procedure that can be used for a system with an Agilent 8753 or 8720 network analyzer.

Preverifying the Device S-Parameter (AC) Data

This procedure uses the s_vgvdf setup to measure the device S-parameters across a calibrated frequency range, with the drain bias voltage swept from 0V to the upper limit of its normal operating range, and several values of gate bias.

The procedure can be used as a quick check that the device performance is good at higher frequencies, before you do a complete data acquisition and model generation. It is not necessary to do the preverification every time you generate a model, especially if you are using data sheet values.

Device Configuration: The input values you set in this procedure configure the device as shown in the following figure.
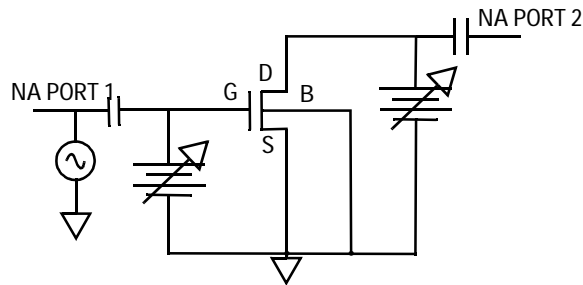
Figure 83   MOSFET Bias Configuration for S-Parameter Preverification

Defining the Instrument States

Select the **Instrument Options** tab and follow the steps described in the section "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system), with the exceptions noted below.

Because you used a frequency list cal rather than a swept frequency cal, note the following:

• Be sure the instrument states you set for the network analyzer correspond to the values you set in your calibration.

• Set Use Fast CW to No.

• Be sure to set the averaging factor the same as in your calibration.

• Set Cal Type [SHN] to H for hardware.

• Set Cal Set No to the cal set where you stored your broadband frequency list calibration.

• Set Use Linear List to Yes.

Then return to this chapter and continue.

Setting the s_vgvdf Inputs

This procedure defines the input signals from both the DC source/monitor and the network analyzer for this measurement. Since this measurement only preverifies the device, the values can be quite conservative and need not stress the device to the limits of its performance. Select the s_vgvdf setup.

Table 27   For the vd input

Set	To
Mode	V
Compliance	no greater than the maximum breakdown current value for the device.
Sweep Type	LIN
Sweep Order	2
Start	0.0
Stop	at or below the upper limit of the device's normal operating range
# of Points	2 or 3 is sufficient to display the data while giving fast results

Table 28   For the vg input

Set	To
Mode	V
Compliance	no greater than the maximum allowable gate current value.
Sweep Type	LIN
Sweep Order	3
Start, Stop, # of Points	to measure at two or three different bias points, in a range from approximately Vgs=Vthreshold to Vgs=3 or 4V.

> **NOTE** The vd and vg ranges apply to an NMOS device. Recognize that for a PMOS device both vd Start and vg Start are negative, and Stop is 0.000.

Table 29   For the freq input

Set	To
Mode	F

Table 29   For the freq input

Set	To
To Node	
From Node	
Unit	
Compliance	
Sweep Type	LIN
Sweep Order	1
Start, Stop, # of Points	correspond with the broadband frequency list calibration you performed.

NOTE   If you set a MHz value, write *MEG* in full.

Measuring and Plotting

NOTE   Before performing the S-parameter measurements, manually take the network analyzer out of hold sweep mode. Press STIMULUS > MENU > MORE > CONTINUAL on an Agilent 8510, or MENU > TRIGGER MENU > CONTINUOUS on an Agilent 8753, to initiate continuous sweep.

1 From the Plots tab, select **Display All**. Two S-parameter plots are displayed.

2 Select **Measure/Simulate > Measure**.

3 Check the S-parameter plots. If the results are unexpected, recheck the values you set in Inputs and Instrument Options. In particular, be sure the instrument states and input stimulus signals match the values you set in the network analyzer broadband calibration.

4 If you change the inputs or instrument states, repeat the measurement. If the data still looks abnormal, the device may

be defective and may need to be replaced with an acceptable one.

5 To save the measured data, select **File > Save As**, type in an appropriate filename (for example *spar_test.set*), and select **OK**.

6 Close the plots.

If you are satisfied with the measured data for your device, continue with the procedures.

Measuring and Extracting Parasitic Resistance Values

This procedure measures the device S-parameters across the frequency range of operation, with the device in an unbiased condition (that is, with both the gate and drain bias set to zero). Under these conditions, the transconductance and drain conductance are negligible, so the equivalent circuit consists only of the parasitic elements and capacitances. The program calculates the intrinsic parasitic resistance from the measured data.

Device Configuration: The input values you set in this procedure configure the device as shown in the following figure.
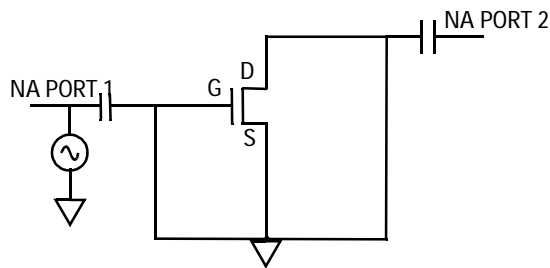
Figure 84   MOSFET Bias Configuration for parasitics Measurement

| NOTE | Make sure you have calibrated the network analyzer before you begin this procedure. Refer to the instructions earlier in this chapter. |

Defining the Instrument States

Perform the same procedure you did for the s_vgvdf measurement. Pay particular attention to the notes listed in that procedure. *The instrument states need to be set independently for each setup.*

Then return to this chapter and continue.

Setting the parasitics Inputs

Most of the input values for this measurement are factory preset and should not be changed.

1 Select the **parasitics** setup.

2 Set the current Compliance for vg and vd to any reasonable values: in this unbiased condition, current values to the device will be negligible.

3 The Sweep Type is set to CONstant for both vg and vd.

4 The vg Value is set to 0 to ground the gate.

5 The vd Value is set to 0 to ground the drain.

6 The freq inputs set the network analyzer for a linear sweep of the calibrated frequency range. Set the freq Start, Stop, and # of Points to correspond with your broadband calibration.

Measuring, Plotting, and Extracting

1 Select **Plots > r_f > Display Plot**.

2 Select **Measure/Simulate > Measure**.

3 When the measurement is complete, select **Extract/Optimize > z > Execute**. The transform calculates the parasitic R values vs frequency from the measured data and enters them in a file named *Para.data*.

4 The r_f plot illustrated shows parasitic resistance with frequency. The resistance data displayed is the real part of the measured device impedance *z.m*.
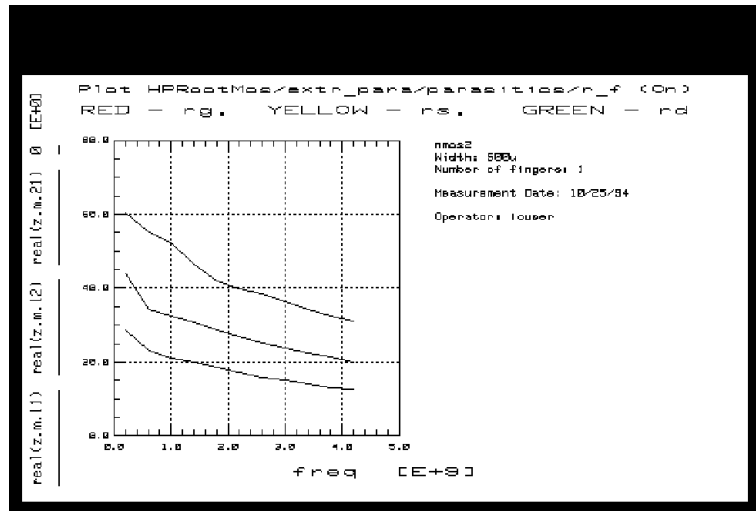
Figure 85   Example parasitics r_f Data

5 If you wish, you can recompute the resistance values over the linear portion of the data only, using the X-low/X-high function as follows:

- Select **Setup Variables**. A variable table will appear, listing X_LOW and X_HIGH with their default values of 0 or current values.

- Choose a relatively linear portion of the plotted traces. Click at the lower and upper boundary points of the linear portion to make a box on the trace, as shown next.
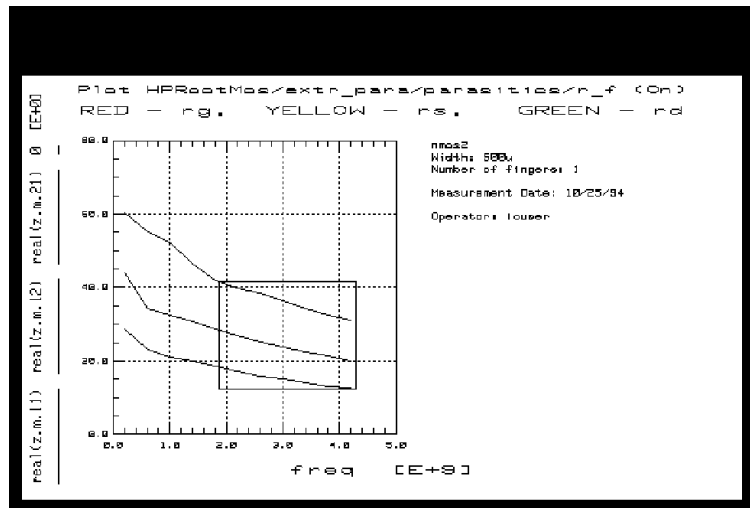
Figure 86   Choosing a Relatively Linear Portion of the Traces

- From the plot menu select **Options > Copy to Variables**. This changes the X_LOW and X_HIGH values in the table to the boundary points you selected.

6 Select **Extract/Optimize > compute_r > Execute**. The transform recalculates the parasitic resistance values as the average of the linear data you defined, and modifies the *Para.data* file accordingly. The values for rs, rd, and rg are listed in the IC-CAP Status window.

7 The parasitic values can be modified after the main data acquisition, just before the model generation.

8 Close the plot.

Setting Up the Main Data Acquisition

This sets up the values for the main measurement procedure, which takes the data from which the model will be generated. The procedure measures S-parameters at a calibrated CW frequency, across a range of drain and gate bias voltages. It takes adaptive measurements, densely spaced in the most nonlinear regions such as at the knees of the I-V curves and at the onset of breakdown, and less densely spaced in the linear

saturation region. The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

Some knowledge of the device characteristics is needed in order to enter valid parameters. The data you measured in the DC and S-parameter preverifications may be useful in determining appropriate values for your device. Do not exceed maximum allowable values. Perform the steps in the following paragraphs.

The data acquisition rate is approximately 700 points per hour. Once you have initiated the transform, the data acquisition takes approximately 1 to 2 hours before your interaction is needed again.

Defining the Instrument States

The instrument states for the main data acquisition are set in the dc_bias and ac_bias setups.

CAUTION   Do not change the Inputs in the dc_bias or ac_bias setups. Their function is not the same as in the other setups. They contain factory-set information essential to the program, and they *must not be changed.*

1   Select the **dc_bias** setup.

2   Select the **Instrument Options** tab to display the DC source/monitor instrument states.

3   Set the Agilent 4142 instrument state the same as for the idvd_vg DC preverification measurement.

4   Select the ac_bias setup, then **Instrument Options**.

5   *Note that the instrument states are different for CW and list frequency measurements.* Set the instrument states *almost* the same as you did for the s_vgvdf preverification measurement, with the following exceptions (because you are now using the CW calibration instead of the broadband frequency list cal):

   • Set Use User Sweep to Yes.

- Set Cal Set No to the cal set or register where you stored your CW calibration.

NOTE

If you fail to use the CW calibration and use the broadband cal instead, the measurement will take a great deal more time than expected. It will measure data at all the bias points at every frequency point. The plotted results will be hard to interpret!

Setting the Variables

You set the values that define the main measurement and the model generation in the data_acquisition and model_generator transforms in the create_mdl setup. Based on your information, IC-CAP characterizes the device within a subset range of the bias plane (Vd-Vg plane). This range is typically larger than the normal safe range for long-term DC device operation specified in the data sheet. In other words, the device is briefly pushed beyond its long-term normal DC operating range. This is required by the model for simulating large-signal AC and transient pulsed operation, where the instantaneous voltages can temporarily exceed the boundaries of long-term safe DC operation.

1 Select the **create_mdl** setup.

2 Select the **Extract/Optimize** tab, and the create_mdl transforms are listed.

3 Select **data_acquisition** to display the variables used in the data acquisition.

Figure 87 Variables in the data_acquisition Transform

4 Follow the steps below to enter variable values appropriate for your device.

5 Do not change the Function name at the top of the table.

6 For Power level, enter the total maximum power dissipation value. This should typically be 2 watts per mm of gate width for devices up to 1 mm.

7 For I Breakdown, enter the reverse gate current breakdown value.

The next four voltage values define hard limits to the measurement domain. The voltage values are set well beyond the normal device operating range to explore the device's boundary response. If the values are set too conservatively, data will not be measured in the nonlinear areas of breakdown and maximum forward conduction. If this occurs in the measurement, return to this transform and change the voltage values. Reduce the minimum values and increase the maximum values.

- Min Vd should typically be slightly negative, to allow data to be taken slightly beyond the bounds of the model so that it is possible to properly extrapolate beyond the data.
- Max Vd should typically be slightly higher than any anticipated breakdown voltage.
- Min Vg should typically be one to several volts lower than Vthreshold for an NMOS device.
- Max Vg should typically be around the maximum gate operating voltage.

8 For Min Step, enter a value to define the step size between densely-spaced measurements.

9 For Max Step, enter a value to define the measurement step size where you expect the data to be relatively linear and you do not need densely-spaced measurements.

10 For Vth, enter the gate threshold voltage Vthreshold.

11 Delta defines a voltage range below Vth where a high density of Vgs measurements is needed due to the nonlinear decrease in drain current at threshold.

12 Eps (Epsilon) is used to predict the next point to be measured, and thus control automatic data acquisition step size. Increasing the value of epsilon decreases the sensitivity of the automatic data acquisition.

13 For Noise thresh, enter a current threshold value for both drain current and gate current below which you do not need closely-spaced measurements. The automatic step-size algorithm is not used below the Noise thresh value, where measured currents are sampled at the maximum step size.

14 For SMU Compl, enter the drain SMU maximum current compliance value in amps. This is the maximum output current capability of the Agilent 4142 plug-in source/monitor unit (SMU) supplying the drain terminal. In the example procedure, this is VD or HPSMU3. For the Agilent 41420A high-power SMU, the compliance is 1A; for the Agilent 41421B medium-power SMU, the compliance is 100 mA.

CAUTION   Also be aware of the current ratings of your bias networks, and do not set current values beyond their specified maximum limits. It is quite possible to destroy the bias networks, and the probes, by applying current values beyond the specified limits. (The standard bias networks for the Agilent 85122A/85123A parameter extraction test systems are rated at 0.5 amp.)

15 You can save the values in the transform to use for another extraction. Select **File > Save As**, then the File Type Transform, type an appropriate filename, and select **OK**. The file is saved with the suffix *.xfm*.

Performing the Data Acquisition Measurement

This procedure performs the measurement you defined. Once you initiate the transform, the data acquisition takes approximately 1 to 2 hours to run before your interaction is needed again. The data acquisition rate is approximately 700 points per hour. The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

1 In the create_mdl setup select the **Extract/Optimize** tab.

2 Select **data_acquisition > Execute**. During the data acquisition process, a running message in the UNIX window indicates the approximate percentage of completion.

CAUTION   Do not perform data acquisition more than once. This will write over the existing data, and will use more time and memory.

3 If you do attempt to perform data acquisition after the measurement has already been done, a dialog box will alert you that a measurement file exists. You then have several choices: If you do not need the data in the current file you can select OK or Return to overwrite it. Or you can abort your new measurement by selecting Cancel. Or if you wish to save the measured data and perform another data acquisition, go to a UNIX window, change directories if

necessary, find the file *Measured.data*, and rename it with another appropriate name, then continue with the data acquisition.

4 Ignore this message if it occurs: **CAUTION: CALSET X STATE MAY BE INVALID.**

5 When the process is complete, a message similar to the following is shown in the UNIX window:

```
INITIAL Extraction Results
Max_num = 66.00
Max_row = 49.00
Done!
```

The numbers are part of the internal function of the program, and irrelevant in user interaction with the program. They are associated with the maximum number of points for a given Vgs, and the number of Vgs steps.

Plotting the Measured Data

The plots in the create_mdl setup provide graphical outputs of the data acquisition and model generation. Most of the plots are of data measured in the main data acquisition. However, Qg and Qd are outputs of the model generator, and will be viewed later.

For hardcopy output, black and white can be reversed for faster printing and easier reading on paper, as explained in the procedure.

If you wish to plot data from a previously stored data set, you will need to perform the steps explained under "To Plot an Earlier Data Acquisition" on the next page. Otherwise, follow these steps:

1 From the Plots tab, select the plot of your choice, and **Display Plot**.

2 You can augment the annotation of a plot by specifying header and footer labels you wish to print with the data. However, it is *essential* not to change the Y data input fields.

3 To change a plot's annotation, go to the Header or Footer lines in the plot definition. You can type a label of your choice in the header or footer input field (right side of the definition

table). Then select **Options > Update Annotation**, and your label is added to the plot.

NOTE   Changing the Y Data input fields can cause anomalies in the measurement and modeling results.

4 If you wish to exchange the black and white in a plot for better hardcopy quality, select **Options** from the plot menu, and **Exchange Black-White** from the pull-down menu. You can also change from colored traces to all black by selecting **Options > Color**. This is particularly useful if one of the traces is a light color such as yellow, which may not show well against a white background.

5 To get a hardcopy of a plot, from the plot menu select **File > Print Setup**. Enter the name of a printer and make any other changes to the printer/plotter setup you wish, then select **OK**.

6 If you have your printer setup completed, select **File > Print**.

To Plot an Earlier Data Acquisition

If you wish to view data from a previously stored Agilent RootMOS data acquisition, you will need to perform these extra steps to bring the data into this model and display the plots.

1 From the IC-CAP main menu select **File > Change Directory**. Type in the path name of the file where your data is stored, and select **OK**.

2 Select the **create_mdl** setup.

3 From the Plots tab, select the plot of your choice, and **Display Plot**. (The displayed plot window will be *blank* and an error message will be displayed. Ignore the error message and click **OK**.) Now select **Display Plot** again, and the plotted data will be properly displayed. (The first time you selected Display Plot, an internal function re-initialized the data from the changed directory.)

The Measured Data Plots

The plots listed next illustrate the data measured by the data_acquisition transform for an example device.

- Id_vd plots the DC measurement of drain current Id as a function of Vds for different values of Vgs. Check this plot to see if you are satisfied with the bias range of the measurement. If not, go back to "Setting the Variables" on page 229 and make appropriate changes in the values of Min Vd, Max Vd, Min Vg, and Max Vg.

- Y11i plots the imaginary part of the Y11 admittance data.

- Y12i plots the imaginary part of the Y12 admittance data.

- Y21r plots the real part of the Y21 admittance data.

- Y22r plots the real part of the Y22 admittance data.

- Vd_Vg plots the distribution of test data in the Vgs-Vds plane. It shows how the data is taken adaptively depending on the device nonlinearities. More test data is taken where the device characteristics change rapidly such as at the knees of the I-V curves and the onset of breakdown, and less in the linear saturation region.

If you are satisfied with the measured data for your device, proceed to the model generation. Be sure the limits of the measurement domain are adequate: the domain should extend beyond the normal operating range of the device. If the measured data does not show the limits of device behavior, you may wish to redefine the voltage ranges you set in the data_acquisition transform, using values that are less conservative, and perform the data_acquisition again.

Generating the Model

This procedure calculates the model parameters from the measured data, and generates a corresponding model for the device. The model nonlinear functions are calculated by contour integration, therefore you must provide the program with a starting point for the contour integration.

If you are generating a model from a previously stored data set, perform the steps listed later in this procedure under "Generating a Model from Previously Measured Data" on page 241.

Computing the Parasitics for the Model Generation

An earlier part of the procedure calculated the intrinsic parasitic resistances of the device and entered them in the file *Para.data*. Now this procedure updates the data and gives you the opportunity to override the calculated values and input values of your own if you believe they are more appropriate.

Select the **main** DUT, and the **DUT Variables** tab. A variable table is displayed, showing current values for the intrinsic parasitic resistances and inductances, as well as other variable values. The following figure illustrates the DUT variables table. Note the intrinsic inductances Ls_tot, Ld_tot, and Lg_tot all have default values of 1.000f.

Name	Value
Rs	32.47
Rd	26.63
Rg	32.19
Ls_tot	1.000f
Ld_tot	1.000f
Lg_tot	1.000f
Max_num	27.00
Max_row	13.00
Vdrain	7.494
Vgate	4.600
Idrain	0.057316
Igate	2.4568E-08
Id_Compliance	133.4m
Ig_Compliance	0.1
MDS_MEASURE_FAS.	1

Figure 88   Variable Table of the main DUT, Showing Parasitic Values

Do not change any of the other variable values in the table besides the R and L values. The other variables listed are internal functions necessary to the program, and can be ignored.

If you have reason to change any of the parasitic values, you can change them in this variable table. For example, if you have calculated the parasitic values by another method, such as from material measurements or by using a different algorithm, you may prefer to use those values. Or based on the model generation results plotted later in the procedure, you may wish to return to this step and modify values, then perform the transforms again.

NOTE   If you make any changes to the parasitic values in the variable table, *do not* perform the compute_para transform.

If you do *not* wish to make changes to the parasitic values, select the **compute_para** transform in the create_mdl setup, and **Execute**. You will probably observe changes in the parasitic values in the table.

NOTE	You can copy the variable table by selecting Detach to display it in a separate window. Resize the window and move it to a convenient place away from the central work area.

Setting the Starting Point for Contour Integration

NOTE	For the modeled nonlinear functions to be valid, contour integration must be done in the normal operating range of the device. Outside the range, the results will be very different and less accurate. The further from the operating range, the less accurate the model will be.

Use the following guidelines to set the starting point of contour integration:

1 The starting point should be set at an intersect of Vd and Vg that is within the saturated class A operating range of the device, where drain voltage is beyond the knee and less than

breakdown, and gate voltage is between Vthreshold and Max Vg.
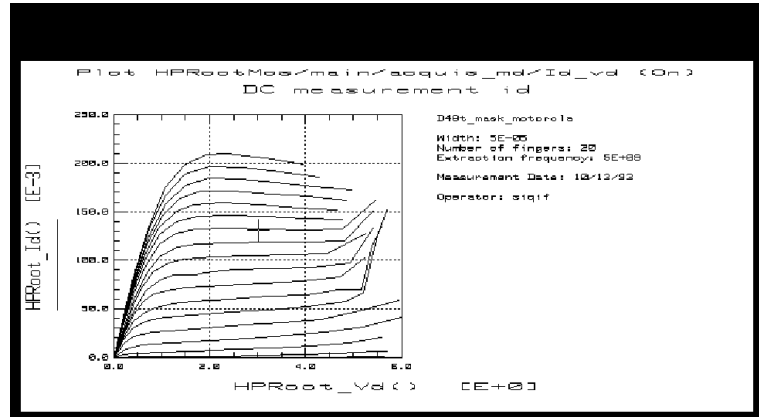
Figure 89   Example of Appropriate Contour Integration Starting Point

2 If you know Vthreshold and Vbreakdown, set:
- Vthreshold < vg_start < vgmax*, and
  Vknee < vd_start < Vbreakdown

or

- vg_start = Vthreshold/2, and vd_start = Vbreakdown/2

(*vgmax is the approximate gate voltage corresponding to the maximum device drain current for a typical drain voltage in saturation. vgmax is generally lower in magnitude than Max Vg.)

Figure 89 shows an appropriate value for the contour integration point of an example device, marked with a + on the plot.

Setting the model_generator Variables

1 Select **create_mdl > Extract/Optimize > model_generator**. The table illustrated below is displayed.

Figure 90 Variables in the model_generator Transform

2 Enter an appropriate value for vd_start. Note that for a PMOS device this will be ≤ 0V.

3 Enter an appropriate value for vg_start. Note that for a PMOS device this will be ≤ 0V.

4 Enter an appropriate value for t_dispersion. This is the dispersion time constant, and its default value is 1e−17 or 10 attoseconds, 10.00a. In most cases, t_dispersion is not a significant factor although it is used in the model generation calculations. However, in a case where you wish to add time dispersion effects, you can increase the value.

Executing the Model Generator Transform

1 Select **Execute** to start the model_generator transform.

2 A dialog box asks you for an MDS file name. The standard file generated by the model for interface with MDS is named *State.mds*. If you want to retain this filename select **OK** or press Enter. Otherwise change the name to a new filename that associates this model with the device, retaining the suffix *.mds* for compatibility with MDS.

NOTE If you plan to do a verification, retain the filename *State.mds* for now, as the verification process requires this filename. You will have another opportunity to change the filename after the verification process.

3 The model generation takes a few seconds. A **Done!** message in the UNIX window indicates the model generation is complete.

Generating a Model from Previously Measured Data

If you wish to generate a model from a previously stored measurement, you will need to perform these additional steps to bring the data into this model in the correct form.

1 From the IC-CAP Main window select **File > Change Directory**. Type in the path name of the file where your data is stored, and select **OK**.

2 Select the **create_mdl** setup.

3 Select **compute_para > Execute**.

4 Now perform the standard procedures described under "Setting the Starting Point for Contour Integration" on page 238 and "Executing the Model Generator Transform" on page 240.

NOTE To plot the data, note the extra steps described under "To Plot an Earlier Data Acquisition" on page 234.

Plotting the Generated Data

In the create_mdl setup, the Qg and Qd plots show the calculated state function data generated by the model_generator transform.

NOTE
The modeled state function data will not be plotted unless the init_parameters macro has been executed (see the instructions near the beginning of this chapter). The icplotnotes file generated by the init_parameters macro provides the x-axis information needed to plot the modeled data. IC-CAP will display an error message if you have not executed the init_parameters macro.

1 From the Plots tab, select the plot of your choice, and **Display Plot**.

- Qg is the distribution of charge current under the gate. The illustration shows this data for an example device.

- Qd is the distribution of charge current under the drain.

Other Outputs of the Model Generation Process

The model generation process results in a number of different output files to be used for different purposes. These include the model file for MDS interface, data on various modeled parameters, measured device data, and files for internal use by the program.

Some files have default filenames assigned by the program. Others may be files you created during the procedure to save device-specific information.

CAUTION
In general, the files should *NOT* be modified, because changing them can cause problems that may invalidate the model. It is also important that you do not change the names of files created by the model generator, with the exception of *State.mds.*

Following is a list of some files that will be or may be created during the modeling process:

- *icplotnotes. This is the only file you can safely edit or add to.* The program appends it as a label to any graphical outputs, to identify the device. It includes a device ID string and any additional information you choose to add to graphs when they are plotted.

- *icplotnumbers*. This is an internal file used by the program to generate a graphical interface. It is related to the Max_num and Max_row messages that appear in the variable table of the main DUT and in the UNIX window at the end of the data acquisition. While these parameters are irrelevant to user interaction with the program, they are essential to its function, and they must appear on-screen. *Do not change this file*.

- *Measured.data*. This is the raw measured S-parameter data as a function of bias and frequency, produced by the create_mdl data acquisition process.

- *Para.data*. This is the S-parameter vs frequency data with the device in an unbiased condition, used by the program to calculate parasitic resistance and inductance values.

- *State.data*. This file is used by the program in plotting nonlinear functions such as Qg and Qd. *Do not change this file*.

- *State.mds*. This is the model file used by the Agilent RF and Microwave Design System (MDS) in a circuit simulation. It is also used in verifying the model, and needs to retain the name *State.mds* in the verification process in order to maintain the MNS simulator link. However, since every Agilent RootMOS model initially has this name, you change the name to identify the device, either when you execute the transform to generate the model (if you are not performing a verification), or at a later time as described at the end of this chapter.

- *Y.data*. This file contains the Y-parameters as a function of bias, to examine through graphical interface. *NO changes* can be made to the *Y.data* file, *except* for adding header and footer labels as explained under "Plotting the Measured Data" on page 233.

- *dc_test.set* (or some other name designated by you). If you decided to save it, this is the measured DC data generated in the idvd_vg DC preverification procedure.

- *spar_test.set* (or some other name designated by you). If you decided to save it, this is the measured S-parameter data generated in the s_vgvdf S-parameter (AC) preverification.

Verifying the Model DC Data

This procedure verifies the model DC data against the DC data measured in the idvd_vg preverification setup, using the MNS simulator link.

NOTE

The simulator used must be MNS. If the *SIMULATOR* in the model variable table is set to another value, change it to *mns*.

The name of the model file must be *State.mds* for the purposes of verification, to maintain the MNS simulator link.

1 Select the **idvd_vg** setup. Make sure the output Type is set to **B** for both measured and simulated data.

2 From the Plots tab, select the **id_vd** plot, and **Display Plot**.

3 Select **Measure/Simulate > Simulate**.
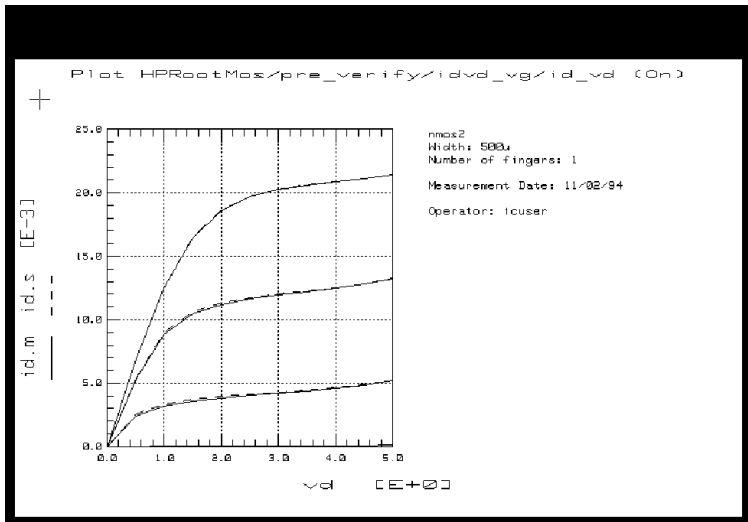
Figure 91 Example Measured and Simulated id_vd Data

4 Check that the simulated data (dotted line) and measured data agree. The illustration shows example measured and simulated id_vd data.

Verifying the Model S-Parameter Data

This procedure verifies the model S-parameter data against the S-parameter data measured in the s_vgvdf setup, using the MNS simulator link.

NOTE

The simulator used must be MNS. If the *SIMULATOR* in the model variable table is set to another value, change it to *mns*.

The name of the model file must be *State.mds* for the purposes of verification, to maintain the MNS simulator link.

1 Select the **s_vgvdf** setup. Make sure the s output Type is set to **B**.

2 Select **Plots > Display All**. Two S-parameter plots are displayed.

- s11_s22 shows example measured and simulated S11 and S22 data

- s12_s21 shows example measured and simulated S12 and S21 data

3 Select **Measure/Simulate > Simulate**. Check the agreement between the simulated data (dotted lines) and measured data.

4 Close the plots.

Storing Your Model and Interfacing with MDS

Once you have developed your device model, you can store it in a file for future use. You can use it for circuit simulation in the Agilent RF and Microwave Design System, inserting it into circuits under design to emulate the characteristics of your device. It can be used to represent any device with the same geometry from the same process, or scaled to represent a different-sized device. It will predict the gain, power, gain compression, and harmonics performance of the device as a component in circuits you design using MDS. It will predict large-signal device operation over a range of bias points and loading conditions.

The file generated by the modeling process for interface with MDS is called *State.mds*. You need to change the name to one that will identify the device under test, since *every* Agilent RootMOS model file is initially named *State.mds*.

If you chose not to perform a verification, you may already have changed the filename using the dialog box presented before the model_generator transform. If not, you can change the filename now in UNIX and store the file as follows:

Storing the Model

1 Use the UNIX directory you created at the beginning of the procedure, when you first opened the model file.

2 From the IC-CAP Main window select **File > Save As**.

3 A dialog box is displayed. Select the desired File Type (for example, *.mdl* for a complete model).

4 Select a new filename that associates this model with the device.

5 Use the UNIX cp command to copy the file *State.mds* to the new filename, including the path name of the designated directory. Select **OK**.

This completes the example Agilent Root MOSFET measurement and model generation procedure. For an explanation of how to use IC-CAP model data in MDS, refer to Appendix D, "Using IC-CAP Data with MDS."

5

Agilent Root Diode Model Generator

This chapter documents the Agilent 85195A Agilent Root diode model generator software. It provides an example procedure that takes you step-by-step through a complete data acquisition and model generation.

The complete procedure includes all the instructions needed to generate a new device model by modifying variable values in factory default tables. After you have set these variables, subsequent procedures for similar devices are much faster.

Model Description

The Agilent Root diode model software is data-based, rather than physical or empirical. It generates a device-specific large-signal model directly from measured DC and small-signal S-parameter data, that accurately represents the terminal characteristics of the device. The model is constructed without circuit simulation or optimization (although simulation may be used in the verification process). Use of S-parameter as well as DC data results in a model that can accurately predict nonlinear device performance. The Agilent Root diode model predicts device performance as a function of bias, power level, and frequency. It allows circuits to be simulated containing devices for which measured data exists but for which there are no good physical or empirical models. The Agilent Root diode model can be used for varactor and Schottky diodes, in a two-port or one-port configuration.

Based on device compliances you provide in software, the data acquisition system calculates the safe operating range for the device. Within this safe operating range, it takes data adaptively at multiple bias points, depending on the specific nonlinearities of the device under test. Measurements are densely spaced in the most nonlinear regions, such as at the onset of breakdown and forward conduction, and less densely spaced in the reverse-biased region.

The data acquisition system sets the DC bias levels and controls the system hardware to take the DC and S-parameter measurements, based on your inputs. The model generator processes the measured data and generates the lookup table required by a circuit simulator. It produces a data file that is directly readable by the Agilent RF and Microwave Design System (MDS). Both IC-CAP and MDS can be run simultaneously in the X Windows environment. During a simulation, the tabular state-function data from the generated Agilent Root diode model is interpolated using spline functions to emulate the terminal characteristics of the device.

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP.

Getting Started in IC-CAP

This example procedure assumes the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System."
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System."
  - Wiltron 360-based system—Refer to your Wiltron documentation

- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.

- The IC-CAP software has been properly installed on the computer.

- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename the SMUs to correspond with the diode terminals

Rename a medium-power SMU such as MPSMU1 as VA (anode supply)

Rename a high-power SMU such as HPSMU3 as VC (cathode supply)

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on opening the IC-CAP software and accessing the model file you need. The default pathname and filename for the Agilent Root diode model file you will use as a first starting point is:

/examples/model_files/diode/HPDiode.mdl

Then return to this chapter and continue with the procedure.

NOTE   Do not install the device yet. The procedure will instruct you to install it after extracting the measurement port resistances.

Before you continue with the IC-CAP procedure, make a new directory in an appropriate location in the UNIX structure, using the mkdir command, to store the data and model files that will be generated. Give the directory a name that will associate it with the device you are going to model. Use the cd command to change to the new directory in UNIX.

MNS Link

The model includes a link to the MNS microwave nonlinear simulator, to allow verification of the model in IC-CAP. The link is set up for MDS version B.07.00, and will need to be modified when MDS is updated. For the link to work, the path and environment variables need to be set correctly: *this is done at the factory for the path and environment of a typical MDS B.07.00 system. Only minor changes to reflect unusual MDS paths or updated MDS versions are likely to be required.* If you do need to modify your UNIX files to reset the MNS link, follow these steps to set the root MDS directory on your system and add $MDSROOT/mdsbin to the path:

For shell or kshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory to automatically set the environment variables:

MDSROOT=/mds_b.07.00 *(depending on the version and location of MDS)*
export MDSROOT
PATH=$MDSROOT/mdsbin: "$PATH"
export PATH

2 Also add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the MNS simulator:

mns mns /mds_b.07.00/hp 85150/lib/mns800 ""
CANNOT_PIPE

For cshell

1 Add the following lines to *.profile* (or *.vueprofile* if you are using HP VUE) in your home directory:

#Setup for mns simulator:

setenv MDSROOT /mds_b.07.00
setenv PATH $MDSROOT/mdsbin: "$PATH"

2 Add a line similar to the following to the *usersimulators* file in */usr/iccap/lib*, so that IC-CAP recognizes the MNS simulator:

mns mns /mds_b.07.00/hp 85150/lib/mns800 ""
CANNOT_PIPE

Using the Agilent Root Diode Model

This chapter provides an example procedure that takes you step-by-step through a measurement and model generation. Much of the data used in this procedure is taken from a diode in a two-port configuration, with some reference to a one-port configuration also. Values you use in generating your own device model will depend on the configuration and the particular characteristics and compliance values for your device: use its data sheet (if available) as a reference.

This example is provided as a starting point. It is a straightforward procedure that uses only a subset of IC-CAP's total capabilities. The complete procedure documented gives all the steps necessary to configure IC-CAP with the parameters for preverification measurements, ideality and parasitic extractions, and the main data acquisition and model generation.

> **NOTE**   Although you will modify the variable values in factory default files during this procedure, do not under any circumstances change the names of any variables or setups. These names are used by the software in the data acquisition and model generation process.

The example procedure gives detailed instructions on how to:

- Initialize device parameters.
- Measure and extract the system measurement port series resistances.
- Make a DC measurement to preverify device DC performance.
- Calibrate the network analyzer.
- Make an S-parameter measurement to preverify device performance at high frequencies.
- Measure and extract the ideality factor.
- Measure and extract the device parasitic resistances and inductances.

- Set up the parameters for the main data acquisition and model generation
- Initiate the main data acquisition. Plot the measured data.
- Initiate the model generation. Plot the calculated state functions.
- Verify the model against measured data.
- Store the data and model files for circuit simulation in Agilent MDS.

Where necessary, other chapters are cross-referenced at appropriate points in the procedure, so that you will have the information you need for system hardware setup and calibration at the proper times.

NOTE	Measurement parameters and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct results.

The procedure provides guidelines on setting values to measure your device. However, the values you use will be based on your knowledge of your device and on information in its data sheet (if available), as well as on the guidelines presented here: they may or may not correspond with the example values.

The illustrations of plotted data are provided as visual examples of possible results. They are not intended to represent a single device, nor do they necessarily correspond with the values given in the procedure. The plots can be used to check the reasonableness of your own data, which should appear similar in shape but not necessarily coincident in values.

The DUT and Setups

The Model window is the central access point for the measurement and model generation procedures. The Model window includes the DUT/Setup panel and the file tabs to access the Model Parameters table, the model variable table, and other IC-CAP features.

The illustration shows the DUT/Setup panel for this model. The DUTs (for example pre_verify), indicated by the inverted triangles, are groupings of similar measurement setups used for related measurements, or to extract related model parameters. The setups (for example i_vs_v) contain the information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.
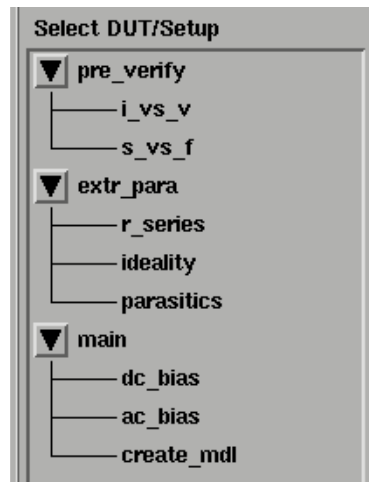
Figure 92 Agilent Root Diode DUT/Setup Panel

The list of macros available from the **Macros** tab provides access to functions that combine certain groups of IC-CAP operations into single operations.

Initializing Device Parameters

You begin the procedure by giving the device an identifier, and entering the configuration (number of ports, two-port or one-port) and device current limits. For this you use the init_val macro and the model variable table. The model uses the current limits to set a safe operating range for the measurements, and measures only within that range. When the init_val macro is

executed, it also initializes internal program functions that are essential in the model generation process, and it provides annotations for the data plots.

1 Click the **Macros** tab to display the names of the macros.

2 In the Select Macro list, select **init_val**, then **Execute**. A dialog box is displayed, asking you to enter a device identifier.

3 Enter a name to identify the device under test, then select **OK** or press Return.

4 The next dialog box asks you to input the device area (that is, the cross-section of the diode) in square meters. The device area obviously will be a very tiny value, perhaps in the region of 200 um*50 um = 1e−8. This area value is required for interface of the finished model with MDS.

5 Another dialog box asks you to enter a directory name. Either select **OK** if you wish to store the model files in the directory shown, or backspace over the directory name and type in the name of another existing directory.

6 The next dialog box asks you to specify the number of device ports.

   • If the device is in a two-port configuration, type 2.

   • If it is one-port, type 1.

   If you try to enter a number other than 2 or 1, a message in the IC-CAP/Status window will tell you the number of ports must be 1 or 2.

7 The macro is executed, and values you selected are entered in the model variable table. Open the model variable table by clicking the **Model Variables** tab.

NOTE   Do not edit the init_val macro. It contains executable code.

Setting the Current Limits

You use the model variable table to set the forward and reverse current limits to be used in measuring your device. To edit the variable table, move the mouse pointer to the appropriate

variable value, click left, and move the pointer over the old information to highlight it. Then type in the new value over the old.

8 For the value of IFwd, enter the maximum forward conducting current for your device. Use the device data sheet, if available, to look up the value.

9 For the value of IBrk, enter the maximum diode breakdown current value (usually less than 1/10 IFwd).

10 Do not change the other values in the table at this time.

Name	Value
Area	1.0
IFwd	30m
IBrk	1m
N	1.140
Number_of_ports	1
SIMULATOR	mns
ANNOTATE_PLOTS	Yes
ANNOTATE_FILE	icplotnotes
ANNOTATE_AUTO	Yes
r1s	0.1
r2s	0.01

Figure 93   Model Variable Table

11 You can display the variable table in a separate window by selecting **Detach**. Resize the window and move it to a convenient place away from the central work area.

Values in the Parameters Table

The figure below illustrates the Model Parameters table, with the initial values for the extrinsic parasitic capacitances, inductance, and resistance. The table is presented so that you can enter values for these extrinsic parasitics based on your knowledge of the device, or on estimates, or on measurements of the open fixture. In addition, you can set limits by entering

minimum and maximum values for each parameter. Parameter values that are outside their limits are clamped to their minimum or maximum values.

Param Name	Value	Min	Max
cac.c	0.000		
cag.c	0.000		
ccg.c	0.000		
lext.l	0.000		
r.r	100.0p		

Figure 94   Model Parameters Table

1 To access the Parameters table, click the **Model Parameters** tab.

2 To change a value in the Parameters table, move the mouse pointer over the current entry and click left to highlight it, then type in a new value.

3 The default for all the c values is 0.000, for both a one-port and a two-port configuration. You can change the value of cac.c for either a one-port or two-port configuration. For a two-port configuration you can also change the values of cag.c and ccg.c. However, the program always uses zero values for cag and ccg in a one-port configuration, even if you manually change the values in the table, since there is no parasitic capacitance from anode to ground or from cathode to ground.

A reasonable capacitance value to enter for a pad on a wafer is 50 fF.

The figure below is a schematic representation of the extrinsic capacitances. Note that the cathode *must* be grounded in the one-port configuration.

one-port configuration   two-port configuration
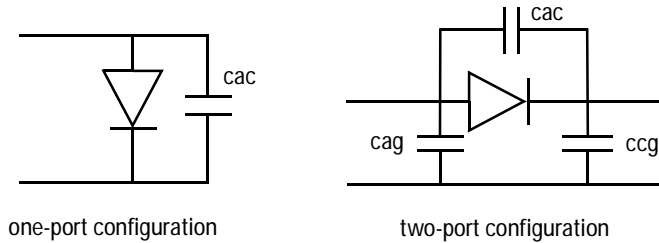
Figure 95   Schematic Diagram of Extrinsic Capacitances

4 lext.l is the bond inductance. The default value is 0.000. If you know or can estimate the bond inductance for your device based on wire length or grid size, enter an appropriate value.

5 r.r is not a real resistance, and is not associated with the measurement system or the diode. It is an artifact used by IC-CAP for verification in a one-port configuration because IC-CAP by default recognizes a two-port configuration. It is set to an initial default value of 10.00 GΩ for a two-port diode configuration. For a one-port configuration, it is set to a very small value such as 100.0 pΩ.

Extracting the Measurement Port Series Resistances

This procedure need be performed *only* following initial installation of the system or if changes are made in the system hardware.

The procedure measures a short circuit at the normal device measurement interface plane. IC-CAP controls the DC source/monitor to force a current and monitor the voltage. The primary measurement is at port 1, and the port 2 measurement is synchronized to the port 1 measurement. You set the measurement parameters by defining the instrument state and the DC input values, as explained in the next pages.

The software calculates the series resistances from the outputs of the bias networks to the inputs of the device under test. The resistance values are then incorporated into the model file, and their effects are removed from the device response during the

modeling process. (The default port resistance for both port 1 and port 2 is 0Ω.) If the diode under test is in a two-port configuration, the series resistance is calculated for both measurement ports; for a one-port configuration, only the port 1 resistance is calculated, and the port 2 resistance is set to 0Ω. (The anode must be connected to port 1.)

Note that the first part of the procedure differs for probe station and in-fixture measurements.

Setting Up for the Measurement

1 If you are using a probe station, position an impedance standard substrate (ISS) on the chuck. For a two-port configuration, place both the port 1 and port 2 probes on a short circuit standard. For a one-port configuration, the placement of the port 2 probe is irrelevant--it need not be lowered.

2 If you are using a fixture, insert the short from the in-fixture calibration kit.

3 From the DUT/Setup panel select the r_series setup.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System."

Set Integ Time to M (medium). Then return to this chapter and continue.

Setting the r_series Input Values

Use the following guidelines to set the inputs (most will not need to be changed from the factory settings):

Table 30   For the ia input

Set	To
Mode	I
To Node	A (anode)
From Node	GROUND
Compliance	as 5 to 10V, because the device is not yet connected
Start, Stop	current values that define a current range within the limits of the bias network
# of Points	a value sufficient to make several measurements across the current range
The Step Size will be set automatically	

- The Unit name must match the name of the SMU connected to the device anode (probably VA). This is also the SMU unit name you set in the hardware window. Refer to the appropriate chapter as follows:

  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

- With a current input, compliance refers to voltage.

CAUTION   The bias networks can be destroyed if IC-CAP measurement current values are set too high. Be sure you know the maximum current ratings of your bias networks, and do not set current values beyond these limits. (The standard bias networks for the Agilent 85122A and 85123A parameter extraction test systems are rated at 0.5 amp.)

Table 31   For the ic input

Set	To
Mode	I
To Node	C
From Node	GROUND
Unit	VC
Compliance	the same as for ia
Sweep Type	SYNC
Ratio	Number_of_ports-1, so that for a two-port configuration the ratio is 1 and a synchronized ic sweep is measured. For a one-port configuration the ratio is 0, therefore no ic sweep occurs and the port 2 series resistance retains the default value of 0Ω
Offset	0.0
Master Sweep	ia, so that the ic sweep is synchronized to the ia sweep

• The Outputs are factory set, and should not be changed.

Measuring and Extracting

1 Under Measure/Simulate select **Measure**.

2 When the measurement is complete, the Measuring light on the DC source/monitor goes out and the IC-CAP wristwatch icon disappears.

3 Select the **Extract/Optimize** tab.

4 Select **compute_rseries > Execute**. The transform calculates the port series resistances from the measured data and enters the values into the model variable table as r1s and r2s.

5 To view a plot of the data, select **Plots > r_i > Display Plot**. The figure illustrates the voltage/current data for a one-port diode configuration. The va/ia (port 1) data is plotted across the measured bias range. The vc/ic (port 2) data has a zero value across the range for a one-port device.
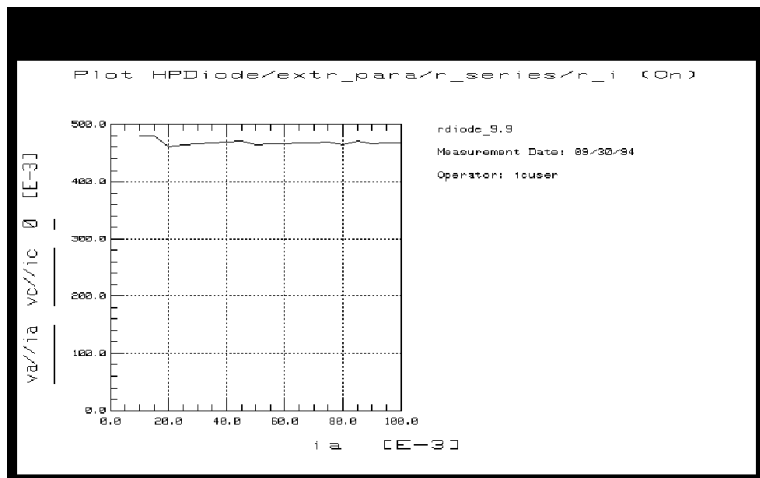
Figure 96   Example r_i Data for a One-Port Configuration

Installing the Device

> **CAUTION**   Ground yourself with an antistatic wrist strap to reduce the chance of electrostatic discharge. (Note that the Agilent 85122 or 85123 system work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.)

Install the diode carefully on the probe chuck or in the fixture. To install the diode in a two-port configuration, connect the anode to port 1 and the cathode to port 2. To install the diode in a one-port configuration, connect the anode to port 1 and the cathode to ground. *A configuration with the anode grounded is NOT supported.* Be sure to handle the device as little as possible, to avoid damaging it. Once the device is in place, avoid bumping the test station.

> **CAUTION**   Remove any high-intensity light sources such as microscope light before taking a measurement. One simple method of blocking light is to place an opaque box on the fixture directly above the device.

Preverifying the Device DC Data

This procedure uses the i_vs_v setup to measure the current values with respect to voltage. It is an opportunity to take a first look at the device performance and explore its characteristics. The procedure starts with conservative values, and iteratively expands the voltage range until it sweeps from breakdown to the voltage where forward conducting current is maximum. This also lets you select the most appropriate measurement range for the main data acquisition and other measurements.

This is a quick way to verify that the device DC characteristics are good, before starting a complete data acquisition and modeling procedure. The data from this measurement can also be used for later comparison with data predicted by the generated model. It is not necessary to do the preverification measurement every time you perform a model generation, especially if you are using a data sheet.

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the r_series measurement. Use the same settings.

Setting the i_vs_v Input Values

This procedure defines the input signals to be applied to the device under test for the DC preverification measurement.

The actual settings you use will be dependent on your device and on the range of measurements you wish to take.

If you are measuring a series of devices of the same type and have already set the input values, go directly to "Measuring and Plotting" on page 265. Otherwise, select the **i_vs_v** setup and use the following settings as guidelines:

Table 32   For the va input

Set	To
Mode	V (voltage)
To Node	A (anode)

Table 32   For the va input

Set	To
From Node	GROUND
Compliance	See note below.
Sweep Type	LIN to provide a linear voltage sweep from start to stop voltage values
Start, Stop	In setting the range, unless you know the operating voltage range of the device, it is best to begin with conservative values to let you observe the device response. Begin by setting a Start value of 0.0 or a slightly negative value. Similarly, begin with a conservative Stop value. If you do know the operating range of the device, you may want to use less conservative values.
# of Points	A value sufficient to clearly display the measurement results.

The Step Size is calculated automatically in software from the start/stop values and number of points.

Table 33   For the vc input

Set	To
Mode	V (voltage)
To Node	C
From Node	GROUND
Unit	VC
Compliance	See note below.
Sweep Type	CONstant
Value	0 to ground the cathode

- The Unit names must conform to the names you set in the hardware window when you configured IC-CAP to recognize the system hardware (probably VA and VC). They must also correspond with the actual SMU connections from the Agilent 4142.

- You provide Compliance values to limit SMU output voltage or current and thus prevent damage to the device under test, as well as to the SMUs, bias networks, and probes, if used. With a voltage input, compliance refers to current: in general, you can set both the va and vc Compliance to IFwd, the forward conducting current value you set in the model variable table.

  However, in setting compliance values, consider this issue: compliances are physically different in the forward and reverse bias ranges. It is important to make sure the compliance is not set too large for breakdown. Although you set only one compliance value for a setup, you are actually sweeping over two ranges, the negative range and the positive range. Two alternatives are possible to overcome this issue: either you can run two setups, one for the forward measurement using IFwd as the compliance value, and one for the reverse measurement using IBrk as the compliance value. Or you can run a single setup, making the va Start value so slightly negative at the first iteration that it will not allow the device to reach breakdown.

- The Outputs are factory set, and should not be changed.

Measuring and Plotting

This procedure triggers the device measurement, controlling the DC source/monitor to supply the signals you defined in the instrument options window and the inputs, and monitors the results.

NOTE Be sure to keep a UNIX window open so that you will see any compliance error that may be reported by the DC source/monitor. If so, you may need to reduce the value of IFwd you set in the model variable table.

NOTE   Before making the DC measurement, manually put the network analyzer in hold sweep mode by pressing STIMULUS > MENU > MORE > HOLD on an Agilent 8510, or MENU > TRIGGER MENU > HOLD on an Agilent 8753.

1 Select **Plots > i_v > Display Plot**.

2 Select **Measure/Simulate > Measure**. When the measurement is complete, the Measuring light on the DC source/monitor goes out, the IC-CAP wristwatch icon disappears, and the measured data is displayed in the plot window.
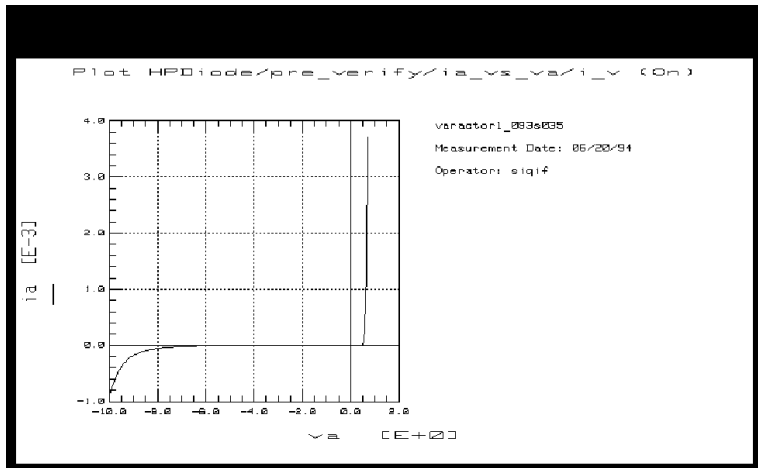
Figure 97   Example DC Measurement of Current vs Voltage

3 Check the plot. The objective of this measurement is to show the device response from breakdown to maximum forward conducting current, as in the illustration. If your plot shows only a partial device response you will need to change the inputs and repeat the measurement, as follows:

4 Go back to the inputs. If necessary, change va Start to a lower value to measure more of the device breakdown characteristics; or change Stop to a higher value to measure the forward conducting current. Select **Measure** again.

5 Iterate as necessary until you have measured a complete device response.

6 If the measured data looks wrong for your device, check the probe contacts, the bias connections, and the measuring instruments. Make sure the device is oriented properly, with the right ports connected to the right terminals (anode to port 1, cathode to port 2 for a two-port device; anode to port 1, cathode to ground for a one-port device). If none of these resolves the problem, the device may be defective.

7 To save the measured data select **File > Save As** and a filename with the suffix *.set* (for example *dc_test.set*). Then select **OK**.

8 Close the plot.

Calibrating the Network Analyzer

It is important to calibrate the network analyzer before you perform any S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and extraction. This procedure uses a frequency list calibration.

Frequency List Mode

Frequency list lets you define an arbitrary list of frequencies in one or more segments. During the measurement sweep the synthesizer is phase-locked at each frequency point, as in the stepped-sweep mode. For the purposes of this model, you will define one frequency segment in the list with a frequency range at least as wide as the operating range of the device. This calibrated swept frequency range will be used for all the S-parameter measurements: the AC preverification, parasitics, and main data acquisition measurements. The model will be generated at one of the calibrated frequencies in the list.

The advantage of frequency list mode for use in a table-based model such as this is that it lets you define a smaller number of measurement points than for a step or ramp frequency cal. This reduces the total measurement time, particularly in the main data acquisition where measurements are taken at all the bias points at each frequency in the list. A # of Points of 5 to 13 provides adequate measurement resolution as well as a quite fast total measurement time for the main data acquisition.

Frequency list calibration procedures for an Agilent 8510-based system and an Agilent 8753-based system are given in the next few pages.

Connecting the Network Analyzer

If you have not already done so, connect the RF cables from the network analyzer test ports to the RF inputs of the bias networks. When it is time to measure the calibration standard devices, connect them to the DC/RF ports of the bias networks.

Frequency List Calibration for an Agilent 8510-Based System

If you are using an Agilent 8510-based system, follow the steps below.

1 On the network analyzer, press LOCAL to gain front panel control. Press PRESET to return to a known standard state.

2 To define a linear frequency list, on the network analyzer in the STIMULUS area press MENU > MORE > EDIT LIST > CLEAR LIST > YES > ADD. A default frequency list segment is presented.

3 To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as M/μ) at the right of the keypad. Similarly, press STOP and set the stop frequency.

<table>
<tr><td>**NOTE**</td><td>Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys in the STIMULUS area of the front panel.</td></tr>
</table>

4 Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range, then press the x1 terminator key. Keep in mind that the fewer the number of points, the less time the measurements take.

5 Press DONE to return to the frequency list menu. The segment you just defined is listed on the screen.

6 Press DONE to return to the main stimulus menu. Press FREQUENCY LIST > SINGLE SEGMENT to activate the

frequency list mode and set the analyzer to sweep the segment you defined.

7 Press MENU > MORE > CONTINUAL to set a continual stimulus sweep.

8 Press MENU > POWER MENU and set the desired power level, then press the x1 terminator key. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If the power level at the sampler goes above −8 dBm, an **IF OVERLOAD** error message is displayed and you will need to reduce the source output power. The default network analyzer power level is 0 dBm.

9 For a device with power drop-off at higher frequencies, you may wish to set a power slope using the stimulus menus. An appropriate power slope would be in the region of 2-3 dB/GHz.

10 In the RESPONSE area, press MENU > AVERAGING ON/restart, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. Although the default is 256, you may wish to use an averaging factor as low as 16 to reduce overall measurement time.

11 If the cal kit constants for your calibration kit are not loaded into the network analyzer, load them from disc now by pressing DISC > LOAD and specifying the data type and file name.

12 If you wish to modify one of the internal cal kit definitions (Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"), do so now.

13 Press CAL > CAL1 or CAL2 (depending on your cal kit), and perform a two-port calibration, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete.

NOTE   IC-CAP does not support a one-port cal: therefore you must perform a two-port cal for either a two-port or a one-port device. If you try to do a one-port cal, IC-CAP will display anomalies and error messages.

14 At the end of the calibration sequence, the cal set numbers are listed in the softkey menu. Press CAL SET 1 to store the calibration in cal set #1. (Or use another available cal set.)

CW Calibration Subset for an Agilent 8510-Based System

1 To create a CW frequency subset of the broadband frequency list calibration, press CAL > CORRECTION ON > CAL SET 1 (this turns on the broadband cal you just completed), then MORE > MODIFY CAL SET > FREQUENCY SUBSET. Use the SUBSET: START and SUBSET: STOP softkeys in the SUBSET menu (*NOT* the front panel keys) to set both start and stop to the same frequency.

2 Press CREATE & SAVE, and store this calibration in cal set #2 or another available cal set.

3 Note the numbers of the cal sets where you save the calibrations: you will need to list one of these numbers as part of the instrument state for *each* of the network analyzer measurements.

NOTE	For more detailed information, refer to the Agilent 8510 *Operating and Programming Manual*.

Frequency List Calibration for an Agilent 8753-Based System

If you are using an Agilent 8753-based system, follow the steps below.

1 On the network analyzer, press LOCAL to gain front panel control. Press PRESET to return to a known standard state.

2 If you are using a system with the 6 GHz receiver option and you wish to measure in the 3 GHz to 6 GHz range, press SYSTEM > FREQ RANGE 3GHz6GHz.

3 To define a linear frequency list, on the network analyzer press MENU > SWEEP TYPE MENU > EDIT LIST > CLEAR LIST > YES > ADD. A default frequency list segment is presented.

4 To change the segment to your chosen frequency range press SEGMENT: START, and use the numerical keypad to set the

start frequency, ending with one of the terminator keys (such as M/μ) at the right side of the keypad. Similarly, press STOP and set the stop frequency.

<table>
<tr><td>NOTE</td><td>Use the SEGMENT: START and STOP softkeys, not the START and STOP hardkeys on the front panel.</td></tr>
</table>

5 Press SEGMENT: NUMBER of POINTS, and enter the number of frequency points to be measured across the range, then press the x1 terminator key. Keep in mind that the fewer the number of points, the less time the measurements take.

6 Press DONE to return to the edit list menu. The segment you just defined is listed on the screen.

7 Press DONE to return to the sweep type menu. Press LIST FREQ > SINGLE SEG SWEEP to activate the frequency list mode and set the analyzer to sweep the segment you defined.

8 Press RETURN > RETURN > TRIGGER MENU > CONTINUOUS to set a continuous stimulus sweep.

9 Press MENU > POWER and set the desired power level, then press the x1 terminator key. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If a receiver input is overloaded (>+14 dBm), the analyzer automatically reduces the output power of the source to −85 dBm and displays the error message **OVERLOAD ON INPUT (R**, **A**, **B) POWER REDUCED**. In addition, the annotation P↓ appears in the left margin of the display to indicate that the power trip function has been activated. When this occurs, reset the power to a lower level, then toggle the SOURCE PWR on/off softkey to switch on the power again.

10 Press AVG > AVERAGING FACTOR, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. A good default averaging factor is 256. To speed your

measurements, you may find it convenient to set an averaging factor as low as 16. Press AVERAGING ON.

11 You can further reduce the noise floor by reducing the receiver input bandwidth. Press IF BW in the average menu, and enter one of the following allowed values in Hz: 3000, 1000, 300, 100, 30, or 10. A tenfold reduction in IF bandwidth lowers the measurement noise floor by about 10 dB; however, the sweep time may be slower. For more information on averaging and the different trace noise reduction techniques, refer to the Agilent 8753 operating manual.

12 Press CAL > CAL KIT, and select the appropriate default or user-defined cal kit for your calibration devices.

13 If you wish to modify an internal calibration kit definition (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"), do so now.

14 Press CAL > CALIBRATE MENU, and perform a two-port calibration at the analyzer front panel, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete. A TRL* or LRM* cal is an appropriate alternative for in-fixture measurements (see Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"). Omit isolation cal. Press DONE, and save the cal in the register number specified in the instrument options table.

15 Detailed procedures for measurements of calibration standards are provided in the Agilent 8753 operating manual.

CW Calibration for an Agilent 8753-Based System

Refer to Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System" for a CW calibration procedure that can be used for a system with an Agilent 8753 or 8720 network analyzer.

Preverifying the Device S-Parameter (AC) Data

This procedure uses the s_vs_f setup to measure the device S-parameters across a calibrated frequency range, at several bias points in the normal operating range of the device. It can use the measured DC preverification data for reference in choosing DC bias points of interest to you.

The procedure can be used as a quick check that the device performance is good at higher frequencies, before you do a complete data acquisition and model generation. It is not necessary to do the preverification every time you generate a model, especially if you are using data sheet values.

Defining the Instrument States

Select the **Instrument Options** tab and follow the steps described in the appropriate chapter shown next (with the exceptions noted below):

- "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Instrument States for Swept Frequency Measurements" on page 624 (Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System."

Because you used a frequency list cal rather than a swept frequency cal, note the following:

- Be sure the instrument states you set for the network analyzer correspond to the values you set in your calibration.

- Set Use Fast CW to No.

- Be sure to set the averaging factor the same as in your calibration.

- Set Cal Type [SHN] to H for hardware.

- Set Cal Set No to the cal set or register where you stored your calibration.

- Set Use Linear List to Yes.

Then return to this chapter and continue.

Setting the s_vs_f Input Values

This procedure defines the input signals from both the DC source/monitor and the network analyzer for this measurement.

1 Select the **s_vs_f** setup.

2 Set the vc inputs the same as for the i_vs_v setup.

3 In the va inputs set Sweep Type to `LIST`, and Sweep Order to `2`.

4 In list sweep mode, IC-CAP lets you specify up to 15 bias voltage values for the measurement. Using your measured DC preverification data as a reference, choose values of interest to you. For example, you might want to view the device response at bias points close to breakdown or in the forward conducting region.

   Be conservative in setting the va Values. Note that the values must be monotonic (set in increasing order, not mixed at random).

5 The number of Values must equal the number of values you enter into the list. (This does not include any 0.000 values at the end of the list.)

6 Set the freq Mode to `F`, Sweep Type to `LIN`, and Sweep Order to `1`.

NOTE	If you set a MHz value, write *MEG* in full. IC-CAP is not case-sensitive, and assumes that *m* or *M* corresponds to *milli*. If an error message is displayed during the measurement that indicates a network analyzer frequency value is too low, you may have entered *M* only.

7 Set the freq Start and Stop values to correspond with the frequency range of your calibration.

8 Set the # of Points to correspond with the calibration. The step size is set automatically.

Measuring and Plotting

Before performing the S-parameter measurements, manually take the
network analyzer out of hold sweep mode. Press LOCAL. Then in the
STIMULUS area press MENU > MORE > CONTINUAL on an Agilent 8510;
or press MENU > TRIGGER MENU > CONTINUOUS on an Agilent 8753, to
initiate continuous sweep.

1 From the Plots tab, **select sp > Display Plot**.

2 Select **Measure/Simulate > Measure**.

3 Check the plot. The S-parameter data displayed should be
   typical for your device. The illustration shows example
   measured S11 data. If your device is in a two-port
   configuration, S22 will also be plotted.

4 If the results are unexpected, recheck the values you set in
   the inputs and instrument options. In particular, be sure the
   instrument states and input stimulus signals match the
   values set in the network analyzer calibration.
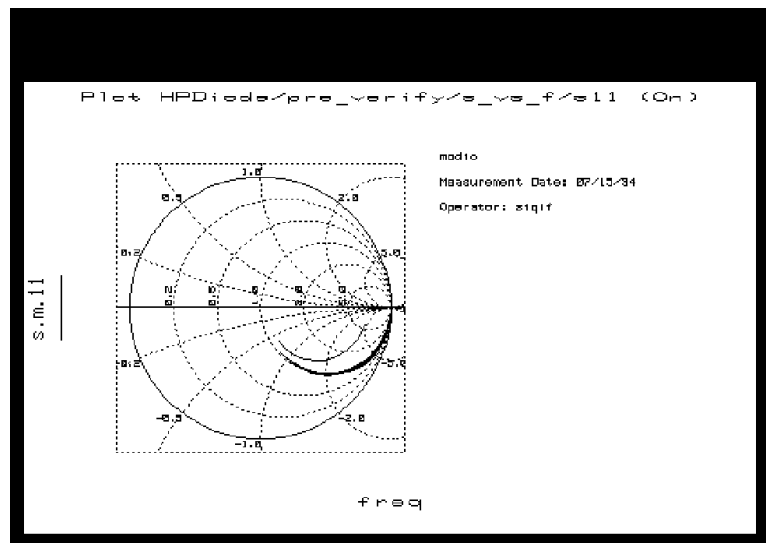
Figure 98   Example Measured S11 Data

5 If you change the inputs or instrument states, repeat the measurement. If the data still looks abnormal, the device may be defective and may need to be replaced with an acceptable one.

6 To save the measured data, select **File > Save As**, type in an appropriate filename (for example *spar_test*), and select **OK**.

If you are satisfied with the measured data for your device, continue with the procedures. Be sure the limits of the measurement domain are adequate: the domain should extend beyond the normal operating range of the device to show the limits of device behavior. If not, you may wish to redefine the voltage range you set in i_vs_v and s_vs_f, using values that are less conservative.

Measuring and Extracting the Ideality Factor

Ideality is the coefficient that multiplies $V_{thermal}$ in the ideal diode equation. The diode resistance is approximated by the equation:

$$R_{diode} \approx \frac{nV_{th}}{I}$$

where n is the ideality factor, V_{th} is $V_{thermal}$ (0.025V at room temperature), and I is the current.

In this setup, the diode is strongly forward biased. The current is swept from a very small positive current value up to maximum, to exercise the diode junction to include the region of its exponential behavior.

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the r_series and i_vs_v measurements. Use the same settings.

Setting the ideality Input Values

Most of the input values for this measurement are factory preset and should not be changed. The current values are based on values you set in the model variable table at the beginning of

the procedure. The other inputs are necessary to set up the required device measurement configuration. Select the ideality setup and use the following settings as guidelines.

Table 34   For the ia input:

Set	To
Compliance	around the same value as the va Stop value used in the i_vs_v DC preverification or a little higher
Sweep Type	LOG scale to allow measurement of the inherently exponential distribution of points
Start and Stop	The purpose of the Start and Stop settings for ia is to force current from ≈0 amps to maximum. Stop is set to IFwd, which is the maximum forward conducting current value you set in the model variable table. Start is set equal to IFwd divided by some large number between 1000 and 10000.
Dec or Oct	Refers to a decade or octave log scale for plotted data. The correct setting for this measurement is D for decade.

- When the ia Sweep Type is set to LOG, the input table automatically includes the fields Dec or Oct and Total Pts in place of Step Size.
- The number of points can be entered as either a # of Points per decade (as in this example) or a Total Pts across the sweep.

Table 35   For the vc input

Set	To
Compliance	See the note below.
Sweep Type	CONstant
Value	0.000, to ground the cathode

- The vc Compliance value of 1.1*IFwd limits the current to a value 10% above IFwd to prevent damage to the device.
- The Outputs are factory set, and should not be changed.

Measuring, Extracting, and Interpreting the Results

This procedure makes the measurement and computes the ideality factor from the results. The procedure explains how to check the measured data and computed ideality, and override the computed value if appropriate.

1 From the Plots tab. Select the **N_vs_va** plot, and **Display Plot**.
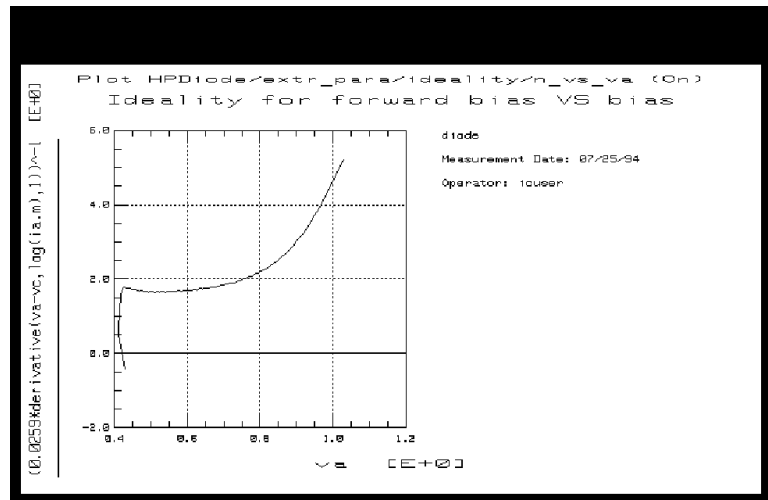
Figure 99   Example Plot of Ideality vs Voltage

2 Select the **Extract/Optimize** tab. Select **N > Execute**. The transform calculates the ideality factor from the measured data and enters the value into the model variables table.

3 To display the table, select the **Model Variables** tab.

4 You can copy the model variables table in a separate window by selecting **Detach**. Resize the window and move it to a convenient place away from the central work area.

5 You can set the program to recompute the ideality factor from only the relatively linear portion of the measured data. Note that the linear region of the measurement is the region where the R_{diode} equation given under "Measuring and Extracting the Ideality Factor" on page 276 applies. The next

steps use the X-low/X-high function to select a linear portion of the trace.

- Select **Setup Variables**. A variable table appears that lists X_LOW and X_HIGH with their default value of 0 or current values.

- Choose a relatively linear portion of the plotted trace. Click the mouse pointer at the lower and upper boundary points of the linear portion, to make a box on the trace as illustrated below.
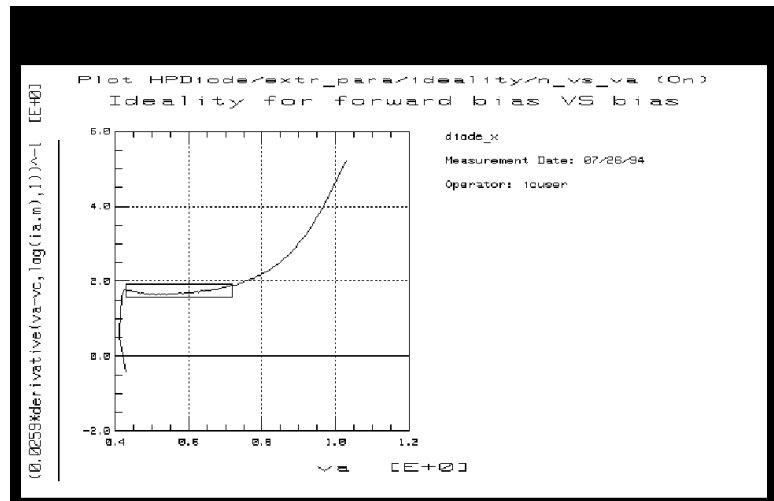
Figure 100   Choosing a Linear Portion of the Plotted Trace

- From the plot menu select **Options > Copy to Variables**. This changes the X_LOW and X_HIGH values in the table to the boundary points you selected.

6   Select **Extract/Optimize > N > Execute**. The transform recalculates the ideality factor as the average of the linear data you defined, and enters the new N value into the model variable table. The new value should be between 1 and 2.

7   If the *N* value in the model variable table is <1, and an out-of-range warning is displayed in the UNIX window, change the limits of the box you defined on the trace and

execute the N transform again. Repeat this until the N value is >1 and < 2.

8 You can check the N value in the variable table against an ideality value on the trace. Put the mouse pointer at a selected point on the linear portion of the trace, click the middle mouse button, and a readout of the Y-axis value at that point will be displayed.

9 If the N_vs_va plot looks significantly different from the illustration, it may be necessary to modify the computed value, as explained in the following steps.

10 If no measured linear region is shown on the trace, a good ideality factor cannot be computed. In this case, you will need to manually enter a value of 1 for N. (In the R_{diode} equation, the NV_{th} term is relatively small compared with I. If a good ideality factor cannot be computed, it is best to eliminate the contribution of the NV_{th} term by using an N value of 1.)

11 To change the N value in the variable table, position the pointer over the current entry and click left to highlight it, then type the new value.

12 You may wish to check the linearity of the device response by viewing the i_vs_va plot, which displays the device current vs voltage on a log scale. The more (log) linear the response, the closer the diode is to being ideal. The illustration gives an example.
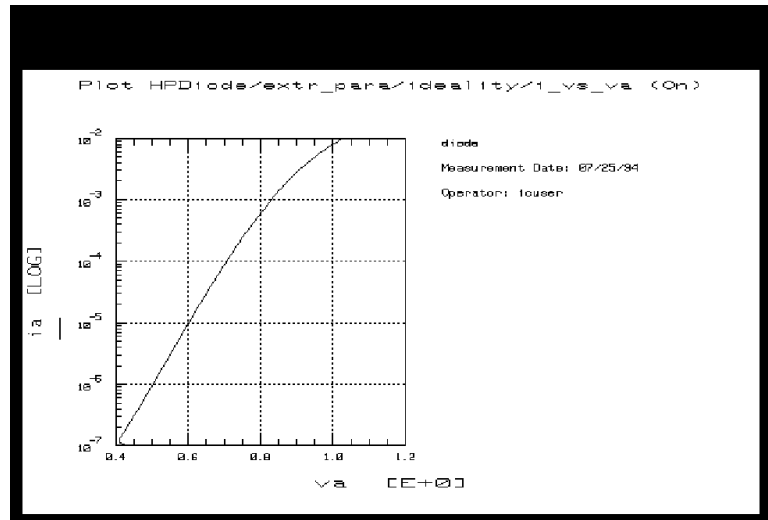
Figure 101   Example Log Scale Plot of Current vs Voltage

13 When you are done, close the plots.

Measuring and Extracting Parasitic R and L Values

This procedure measures S-parameters vs frequency, with the diode forward-biased and a fixed value of forward conducting current. The program calculates the intrinsic parasitic resistance (r) and inductance (l) values from the measured data. The illustration below is a simplified schematic representation of the intrinsic r and l values. In calculating r and l the program takes into account the extrinsic parasitic capacitance, inductance, and resistance values entered in the Parameters table at the beginning of the procedure.
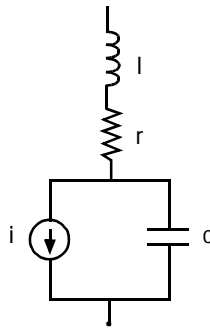
Figure 102   Simplified Schematic Diagram of Intrinsic r and l

NOTE   Make sure you have calibrated the network analyzer before you begin this procedure. Refer to the instructions earlier in this chapter.

Defining the Instrument States

Perform the same Instrument Options procedure you did for the s_vs_f measurement. Pay particular attention to the notes listed in that procedure. *The instrument states need to be set independently for each setup.*

Setting the parasitics Input Values

Most of the input parameters for this measurement are factory preset and should not be changed. The current values are based on values you set in the model variable table at the beginning of the procedure. The other inputs are necessary to set up the required device measurement configuration.

1 Select the **parasitics** setup.

2 Set the ia voltage Compliance to around the same value as the a Stop value used in the i_vs_v DC preverification, or slightly higher.

3 The ia Sweep Type is set to CONstant with a Value of IFwd to drive the diode with a constant maximum forward conducting current.

4   The vc Sweep Type is set to CONstant with a value of 0.000, to ground the cathode.

5   The vc Compliance value of 1.1*IFwd limits the current to a value 10% above IFwd to prevent damage to the device.

6   The freq inputs set the network analyzer for a linear sweep of the calibrated frequency range. Set the freq Start and Stop values to correspond with the frequency range of your calibration.

7   Set the # of Points to correspond with the calibration.

Measuring, Plotting, and Extracting

1   Select **Plots > Display All**.

2   Select **Measure/Simulate > Measure**. The program automatically calculates the r and l values vs frequency from the measured data and enters them in a file named Para.data. (The z transform is automatically executed after the measurement.)

The plots display the parasitic r and l as a function of frequency. The resistance data displayed is the real part of the measured impedance z.m. The inductance data is the imaginary part of z.m divided by $2\pi f$.

The parasitics will be recalculated later at the frequency of extraction only.

3   Close the plots.

The Main Data Acquisition

This is the main measurement procedure that takes the data from which the model will be generated. The procedure measures S-parameters over a calibrated microwave frequency range, across a range of bias voltages. It takes adaptive measurements, densely spaced in the most nonlinear regions such as at the onset of breakdown and the knee of the forward conducting curve, and less densely spaced in the reverse-bias region (except where the capacitance data changes rapidly).

The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

Preparing

Some knowledge of the device characteristics is needed in order to input valid parameters. Follow the steps below to input appropriate values for your device. Do not exceed maximum allowable values. (If a data sheet is available for the device, refer to it for specifications and compliances.)

Defining the Instrument States

The instrument states for the data acquisition are set in the dc_bias and ac_bias setups.

CAUTION	Do not change the inputs in the dc_bias or ac_bias setups. Their function is not the same as in the other setups. They contain factory-set information essential to the program, and they *must not be changed*.

1 Select the **dc_bias** setup.

2 Select the **Instrument Options** tab to display the DC source/monitor instrument states.

3 Set the Agilent 4142 instrument state the same as for the i_vs_v DC preverification and the *ideality* measurement.

4 Select the **ac_bias** setup.

5 Set the instrument states the same as for the s_vs_f preverification and the parasitics measurement (otherwise the calibration will be invalid).

Defining the Voltages and Step Sizes for the Measurement

1 Select the **create_mdl** setup.

2 Select **Extract/Optimize > acquire_data**. The table illustrated below is displayed.
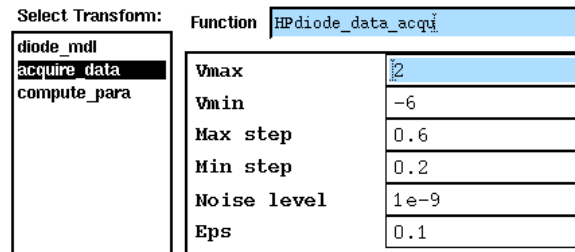
Figure 103   Opened acquire_data Transform in create_mdl Setup

3 Edit the variable values in the transform by entering values appropriate for your own device.

4 For the value of Vmax, enter the maximum allowable forward voltage.

5 Enter a value for Vmin that corresponds to the reverse breakdown voltage.

6 Enter a value you choose for Max step. This defines the measurement step size for regions where you expect the data to be relatively linear and measurements to be more widely spaced.

7 Enter a value you choose for Min step. This defines the step size for densely-spaced measurements where the data is most nonlinear.

8 Enter a value for Noise level. This is the current threshold value below which the automatic step-size algorithm is not used. Set a value below which you do not need closely-spaced measurements, as measured currents below this value are sampled at the maximum step size.

9 Enter a value for Eps (epsilon). Epsilon is used to predict the next point to be measured and thus control automatic data acquisition step size. Increasing the value of epsilon decreases the sensitivity of the automatic data acquisition. Epsilon should generally be between 0.1 and 0.3; the default value is 0.1.

10 If you wish to save these transform values for use with similar devices, select **File > Save As**. Then select the File Type Transform, type in a filename that conforms to UNIX

requirements, and select **OK**. The file is saved with the suffix *.xfm* for transform.

The create_mdl Inputs

Don't change the inputs. The create_mdl setup, like the other setups, includes an input configuration list. However, the function served by this list is not the same as in the other setups. It is factory-set internal information used by the program, and is essential to its function.

Performing the Data Acquisition Measurement

This procedure performs the measurement you defined. Once you initiate the transform, the data acquisition procedure takes approximately 10 to 20 minutes. The data acquisition rate is approximately 10 points per minute. The number of data points measured depends on the nonlinear behavior of the device and on the measurement conditions you set up.

Measuring and Extracting

1 In the create_mdl setup select the **Extract/Optimize** tab.
2 Select **acquire_data > Execute**. During the data acquisition process, a running message in the UNIX window indicates the approximate percentage of completion.

CAUTION   Do not perform data acquisition more than once. This will write over the existing data, and will use more time and memory.

3 If you do attempt to perform data acquisition after the measurement has already been done, a message will ask you **DO YOU WISH TO OVERRIDE?**. You then have several choices: If you do not need the data in the current file you can select Return or OK to overwrite it. Or you can abort your new measurement by selecting Cancel. Or if you wish to save the measured data and perform another data acquisition, go to the UNIX window, change directories if necessary, and rename the file *Measured.data* with another appropriate name, then continue with the data acquisition.

4 Ignore this message if it occurs: **CAUTION: CALSET X STATE MAY BE INVALID**.

5 When the process is complete, a message similar to the following is displayed in the UNIX window:

```
INITIAL Extraction Results
Pts_num = 66.00
Done!
```

The numbers are part of the internal function of the program, and irrelevant in user interaction with the program. They are associated with the number of points in the measurement.

Plotting the Measured Data

The plots in the create_mdl setup provide graphical outputs of the data acquisition and model generation. For hardcopy output, black and white can be reversed for faster printing and easier reading on paper, as explained in the procedure.

If you wish to plot data from a previously stored data set, you will need to perform the steps explained under "To Plot an Earlier Measurement" on page 288. Otherwise, follow these steps:

Viewing and Printing the Plots

1 From the Plots tab, select the plot of your choice, and **Display Plot**.

2 You can augment the annotation of a plot by specifying header and footer labels you wish to print with the data. However, it is *essential* not to change the Y data input fields.

3 To change a plot's annotation, go to the Header or Footer lines in the plot definition. You can type a label of your choice in the header or footer input field (right side of the definition table). Then select **Options > Update Annotation**, and your label is added to the plot.

NOTE	Changing the Y Data n input fields can cause anomalies in the measurement and modeling results.

4 If you wish to exchange the black and white in a plot for better hardcopy quality, select **Options** from the plot menu, and **Exchange Blk Wht** from the pull-down menu. You can also change from colored traces to all black by selecting **Options > Color**. This is particularly useful if one of the traces is a light color such as yellow, which may not show well against a white background.

5 To get a hardcopy of a plot, from the plot menu select **File > Print Setup**. Enter the name of a printer and make any other changes to the printer/plotter setup you wish, then select **OK**. Select **File > Print**.

To Plot an Earlier Measurement

If you wish to view data from a previously stored Agilent Root diode measurement, you will need to perform these extra steps to bring the data into this model and display the plots.

1 From the IC-CAP main menu select **File > Change Directory**. Type in the pathname of the file where your data is stored, and select **OK**.

2 Select the **create_mdl** setup and select the **Plots** tab.

3 Select the plot of your choice, and **Display Plot**. (The displayed plot window will be blank and an error message will be displayed.) Ignore the error message and click **OK**.) Now select **Display Plot** again, and the plotted data will be properly displayed. (The first time you selected Display Plot, an internal function re-initialized the data from the changed directory.)

Generating the Model

This procedure calculates the model parameters from the measured data, and generates a corresponding model for the device.

If you are generating a model from a previously stored data set, perform the steps listed later in this procedure under "To Generate a Model Using Data from an Earlier Measurement." Otherwise, follow these steps:

Defining the Extraction Frequency

The model is extracted at a single frequency, which should be approximately the frequency at which the device will be used. It *must* be one of the measured points from the calibrated frequency list. To determine a frequency for the extraction, use a calibrated measurement point equal to the start frequency plus an integer multiple of the step size. For example:

200 MHz (start freq) + 1 · 4.45 GHz (step size) = 4.65 GHz

1 In the create_mdl setup select the **Setup Variables** tab. The variables table illustrated is displayed.

Name	Value
Freq	4.65G
R	4.832
L_tot	97.42p
X_LOW	-377.0m
X_HIGH	49.59m

Figure 104 Variable Table of the create_mdl Setup

2 Manually change the value of Freq to your chosen extraction frequency.

Computing the Parasitics at the Extraction Frequency

1 Select **Extract/Optimize > compute_para > Execute**. This calculates the parasitic r and l values at the extraction frequency and enters them in the create_mdl variables table,

in place of the average value of the parasitics over the total measured frequency range.

Scaling the i Plot Before Generating the Model

Sometimes excessive noise may be present in the IV characteristics near zero, although this is not generally visible with standard scaling. To prevent the model generation process from interpolating data points in this noisy region, the model can smoothly join the data outside this region and connect it across the noisy region.

The following steps use the measured data in the i plot to determine if there is excess noise near zero, and to curve-fit the measured data if necessary.

1 Select **Plots > i > Display Plot**.

2 To obtain a closer view of the measured data around the knee of the forward conducting current curve, click trace points to the left and right of zero to redefine the plot range. Then in the plot menu select **Options > Rescale** to expand the plot to the new range.

3 Perform the last step repeatedly until you can see if there is low-level noise on the trace.

If the current appears smooth even at expanded resolution, go to "Executing the Transform" on page 292.
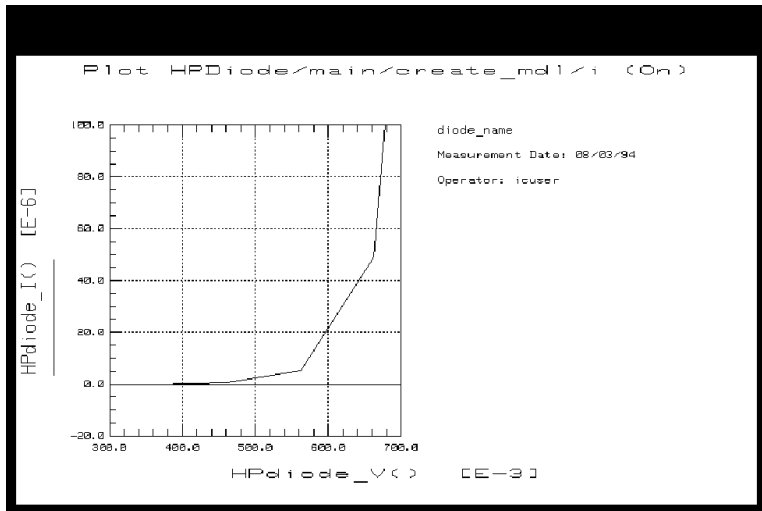
Figure 105   i Plot Expanded to Show Data Near Zero

If the current appears noisy at expanded resolution, use the X-low/X-high function for curve-fitting, as follows:

• Click the mouse at the left and right boundary points of the noisy portion of the trace, to make a box on the trace.

• From the plot menu select **Options > Copy to Variables**. This sets the X-LOW and X-HIGH values in the create_mdl setup variable table to the boundary points you selected. It also sets the values for LowBound V and HighBound V in the diode_mdl transform.

• Note that the value for X-low should be slightly negative and for X-high should be slightly positive. An error message will be displayed if the values are inappropriate.

• Now when the model is generated, the noisy data will be averaged over the X-low/X-high range you defined.

4 Close the i plot.

Executing the Transform

1 Select the **Extract/Optimize** tab.

2 Select **diode_mdl > Execute** to start generating the model for the device under test.

3 A dialog box asks you for an MDS file name. The standard file generated by the model for interface with MDS is named *diode.mds*. If you want to retain this filename select **OK** or press Enter. Otherwise change the name to a new filename that associates this model with the device, retaining the suffix *.mds* for compatibility with MDS.

<table>
<tr>
<td>

NOTE
</td>
<td>

If you plan to do a verification, retain the filename *diode.mds* for now, as the verification process requires this filename. You will have another opportunity to change the filename after the verification process.
</td>
</tr>
</table>

4 A **Done!** message in the UNIX window indicates when the model generation is complete.

Generating a Model from Previously Measured Data

If you wish to generate a model from a previously stored measurement, you will need to perform this additional step to bring the measured data into this model.

1 From the IC-CAP main menu select **File > Change Directory**. Type in the pathname of the file where your data is stored, and press Return.

2 Now perform the standard procedure, "Generating the Model," described above.

<table>
<tr>
<td>

NOTE
</td>
<td>

For plotting the data, note the extra steps described under
</td>
</tr>
</table>

Plotting the Generated Data

1 To view a specific plot select the plot and **Display Plot**. The c plot displays the high-frequency intrinsic capacitance as a

function of voltage. The following figure shows capacitance data for an example device. The response is properly flat below pinch-off for this FET-like diode, with more points taken where the capacitance changes faster. The data is extrapolated in the forward region where forward bias capacitance is not extractable.
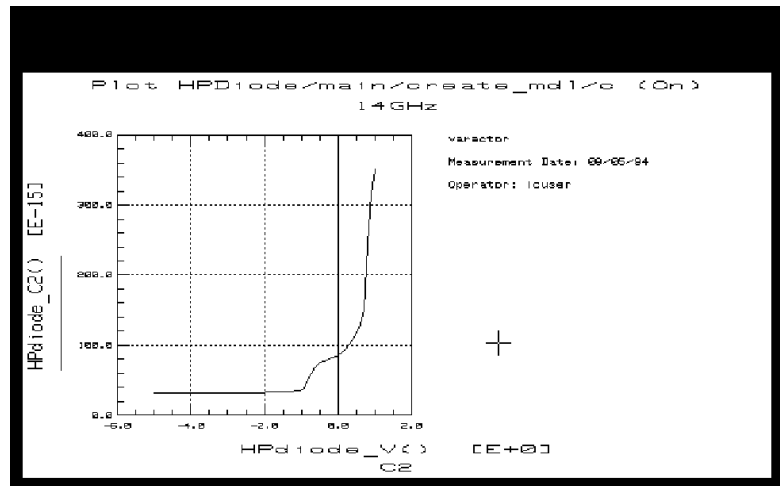
Figure 106   Example Capacitance Data

NOTE   Capacitance characteristics of different diodes may be extremely different.

2 The compare_i plot displays the agreement between the modeled and measured current. When agreement is good, it may be difficult to distinguish the extracted data (yellow trace) from the measured data (red trace). To expand the trace resolution, from the plot menu select **Options > Rescale**. The data illustrated in the following figure shows good agreement even with expanded resolution.
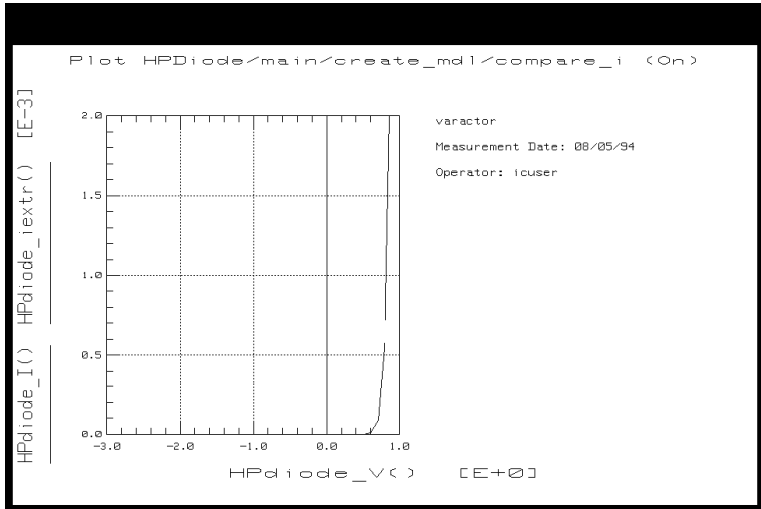
Figure 107   Example compare_i Data

3 Close the plots.

Other Outputs of the Model Generation Process

The model generation process results in a number of different output files to be used for different purposes. These include the model file for MDS interface, data on various modeled parameters, measured device data, and files for internal use by the program.

Some files have default filenames assigned by the program. Others may be files you created during the procedure to save device-specific information.

CAUTION   In general, the files should *NOT* be modified, because changing them can cause problems that may invalidate the model. It is also important that you do not change the names of files created by the model generator, with the exception of *diode.mds*.

Following is a list of some files that will be or may be created during the modeling process:

- *icplotnotes. This is the only file you can safely edit or add to.* The program appends it as a label to any graphical outputs, to identify the device. It includes a device ID string and any additional information you choose to add to graphs when they are plotted.

- *icplotnumbers*. This is an internal file used by the program to generate a graphical interface. It is related to the Pts_num message that appears in the UNIX window at the end of the data acquisition. While the Pts_num parameter is irrelevant to user interaction with the program, it is essential to its function, and must appear on the screen. *Do not change this file*.

- *Measured.data*. This is the raw measured S-parameter data as a function of bias and frequency, produced by the create_mdl data acquisition process.

- *Para.data*. This is the S-parameter vs frequency data at forward bias, used by the program to calculate parasitic resistance and inductance values.

- *Diode.data*. This file is used by the program in plotting nonlinear functions. *Do not change this file*.

- *Diode.mds*. This is the model file used by the Agilent RF and Microwave Design System (MDS) in a circuit simulation. It is also used in verifying the model, and needs to retain the name *Diode.mds* in the verification process in order to maintain the MNS simulator link. However, since every Agilent Root diode model initially has this name, you change the name to identify the device, either when you execute the transform to generate the model (if you are not performing a verification), or at a later time as described at the end of this chapter.

- *dc_test.set* (or some other name designated by you). If you decided to save it, this is the measured DC data generated in the DC preverification procedure.

- *spar_test.set* (or some other name designated by you). If you decided to save it, this is the measured S-parameter data generated in the S-parameter (AC) preverification.

Verifying the Model DC Data

This procedure verifies the model DC data against the DC data measured in the i_vs_v setup, using the MNS simulator link.

> **NOTE**
> The simulator used must be MNS. If the SIMULATOR in the model variable table is set to another value, change it to mns.
>
> The name of the model file must be *Diode.mds* for purposes of verification, to maintain the MNS simulator link.

1 Select the **i_vs_v** setup. Make sure the output Type is set to **B** for both measured and simulated data.

2 From the Plots tab, select the **i_v** plot, and **Display Plot**.

3 Select **Measure/Simulate > Simulate**.

4 Check that the simulated data (dotted line) on the plot is in good agreement with the measured data.

Verifying the Model S-Parameter Data

This procedure verifies the model S-parameter data against the S-parameter data measured in the s_vs_f setup, using the MNS simulator link.

> **NOTE**
> The simulator used must be MNS. If SIMULATOR in the model variable table is set to another value, change it to mns.
>
> The name of the model file must be *Diode.mds* for the purposes of verification, to maintain the MNS simulator link.

1 Select the **s_vs_f** setup. Make sure the output Type is set to **B**.

2 Select **Plots > sp > Display Plot**.

3 Select **Measure/Simulate > Simulate**.

4 Check that the simulated data (dotted lines) on the plot is in good agreement with the measured data.

Generating a Model at Another Frequency

You may wish to generate a model at a different frequency, which must be another measured point from the calibrated frequency list (see "Defining the Extraction Frequency" on page 289). (During the data acquisition, data was taken at each of the calibrated frequencies in the frequency list.) If so, follow these steps:

1 If you are satisfied with the model you have just generated, copy the file *Diode.mds* to another filename. *(Be sure to use a different name for the new model than you used for the first one.)*

2 Select the **create_mdl** setup.

3 Select the **Setup Variables** tab to open the create_mdl variable table.

4 Change the value of Freq to the new calibrated frequency of your choice.

5 Select **Extract/Optimize > compute_para > Execute** to calculate the parasitic r and *l* values at the new extraction frequency.

6 Select **diode_mdl > Execute** to start generating the new model. Unless you wish to perform a verification afterwards, enter another filename, possibly one that includes the new extraction frequency.

7 When the **Done!** message appears in the UNIX window, observe the plots.

8 If you wish, verify the model against the measured DC and S-parameter data using the previous procedures.

Storing Your Model and Interfacing with MDS

Once you have developed your device model, you can store it in a file for future use. You can use it for circuit simulation in the Agilent RF and Microwave Design System, inserting it into circuits under design to emulate the characteristics of your device. It can be used to represent any device with the same geometry from the same process, or scaled to represent a different-sized device. It will predict the gain, power, gain compression, and harmonics performance of the device as a

component in circuits you design using MDS. It will predict large-signal device operation over a range of bias points and loading conditions.

The file generated by the modeling process for interface with MDS is called *Diode.mds*. You need to change the name to one that will identify the device under test, since *every* Agilent Root diode model file is initially named *Diode.mds*.

If you chose not to perform a verification, you may have already changed the filename using the dialog box presented before the diode_mdl transform. If not, you can change the filename now in UNIX and store the file as follows:

Storing the Model

1 Use the UNIX directory you created at the beginning of the procedure, when you first opened the model file.

2 From the IC-CAP Main window select **File > Save As**.

3 A dialog box is displayed. Select the File Type (*.mdl* for a complete model) by clicking on the corresponding radio button.

4 Select a new filename that associates this model with the device.

5 Use the UNIX cp command to copy the file *Diode.mds* to the new filename, including the pathname of the designated directory. Select **OK**.

This completes the example Agilent Root diode measurement and model generation procedure. For an explanation of how to use IC-CAP model data in MDS, refer to Appendix D, "Using IC-CAP Data with MDS."

6
Agilent EEFET3/EEHEMT1 Model

The Agilent 85192B EEFET3/EEHEMT1 model is an empirical analytic model developed for the purpose of fitting measured electrical behavior of GaAsFETs and HEMTs. The model equations were developed concurrently with the parameter extraction techniques to ensure the model contains only parameters that are extractable from measured data. The model can be automated with extraction macros, or the parameters can be extracted individually from measured data.

The model includes the following features:

- An accurate isothermal drain-source current model that fits virtually all processes.

- Self-heating correction for the drain-source current.

- A charge model that accurately tracks measured capacitance values.

- A dispersion model that permits simultaneous fitting of high-frequency conductances and DC characteristics.

- A breakdown model that describes gate-drain current as a function of both Vgs and Vds.

- The capability to extrapolate outside the measurement range used to extract the model.

Agilent Technologies

Getting Started in IC-CAP

The starting point of this tutorial assumes the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"
  - Wiltron 360-based system—Refer to your Wiltron documentation
- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.
- The IC-CAP software has been properly installed on the computer.
- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter, listed below:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

  Rename SMU1 (for medium power)

  Rename SMU2 (for high power)

Then return to this chapter and continue with the procedures here.

/examples/model_files/mesfet/HPEEfet3.mdl

Before you continue with the IC-CAP procedures, make a new directory (using the mkdir command), to store the data and model files that will be generated. Give the directory a name that will associate it with the devices you are going to model. Use the cd command to change to the new directory in UNIX.

NOTE	The model file *HPEEhemt1.mdl* is the same model as *HPEEfet3.mdl*. The only difference is that *HPEEfet3.mdl* includes premeasured device data from a MESFET, and *HPEEhemt1.mdl* includes premeasured device data from a HEMT.

Using the EEFET3/EEHEMT1 Example Model Files

This chapter provides a sequence of example procedures that take you step-by-step through a FET or HEMT measurement and extraction. The procedures are arranged in the proper sequence to allow earlier measured data or extracted parameters to be used as a basis for later measurements or extractions. The extraction is the same for a FET or a HEMT, except that the additional parameters used for a HEMT can be omitted in a FET extraction.

A series of different IC-CAP setups are used to measure current, voltage, or S-parameters vs. bias under different bias and frequency conditions. The model parameters are then extracted from the various sets of measured data.

NOTE   These procedures measure the device in a common-source configuration.

Macros automate the modeling procedure so that a series of similar devices can be modeled quickly with minimal user intervention, although you can execute individual transforms for specific measurement and parameter extractions.

The detailed procedures (executing individual transforms) in this chapter include instructions for the following:

- Performing a set of device preview measurements and setting measurement variable values based on the results.

- Setting the hardware instrument states and input variables for each measurement.

- Making the parasitic, DC, and S-parameter measurements: qualifying the results.

- Extracting and optimizing the individual model parameters.

- Verifying the extraction results against the measured data.

- Storing the model.

Where appropriate, other chapters are cross-referenced, so that all system hardware setups and calibrations are performed at the right times.

NOTE Measurement parameters and measured data are specific to individual devices. Information and data are provided here as examples and guidelines, and are not intended to represent the only correct method or results.

The procedures start with example values for the model variables. They provide guidelines on setting the values to measure your own devices. However, the values you use will be based on your knowledge of your FET/HEMT devices or on information provided by the device vendors, as well as on the guidelines presented here: they may or may not correspond with the examples. In some cases, it may be necessary to set the variable values for the preview measurements, make a measurement, and then reset the variables based on the measurement results. The procedures give suggestions on judging the applicability of the variable values and the measured data.

The illustrations of plotted data are provided as visual examples of possible measurement and simulation results. They are not intended to represent a sequential measurement and extraction of a single device, nor do they necessarily correspond with the example measurement inputs and variable values in the procedures. The plots can be used to check the reasonableness of your own data, which should appear similar in shape but not necessarily coincident in values.

In addition, the IC-CAP software itself includes measured data for some of the setups. This data shows additional examples of possible results. Many, though not all, of the input settings for this measured data correspond with the example settings in this manual.

The DUTs and Setups

The DUT/Setup panel is part of the Model window, which is the central access point for all the individual measurement and modeling procedures.The Model window includes tabs to access the macros, the Model Parameters table, and other IC-CAP features. To open this model, click the **Examples** button on the toolbar and locate */model_files/mesfet/HPEEfet3.mdl*.
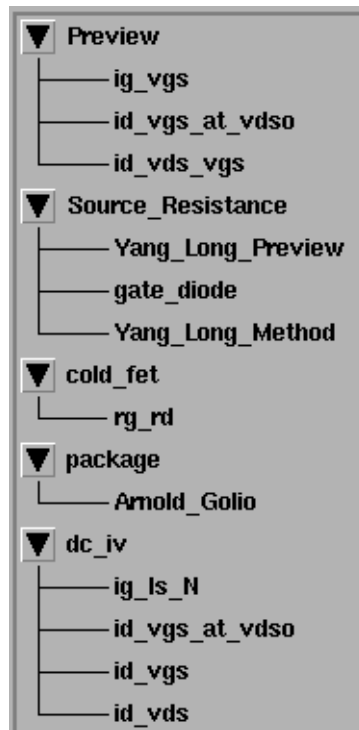
Figure 108   Portion of the Agilent EEFET3/EEHEMT1 DUT/Setup Panel

In the DUT/Setup panel the DUTs (for example Preview)—indicated by the inverted triangles—are groupings of similar measurement setups used for related measurements and extractions. The setups (for example ig_vgs) contain the

information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.

The measurement and parameter extractions are most easily performed with macros. If you use macros, you will not need to use the individual setups. However, if you want to extract individual parameters or "fine-tune" a parameter, you will use the individual setups. (Note that you can also write macros to control other repetitive tasks, using IC-CAP's Parameter Extraction Language, PEL.)

ReadMe is not actually a macro, but an abbreviated version of the instructions in this chapter. It is most appropriate for users who are somewhat familiar with the model. You can view the *Readme* file in a separate window by clicking Detach. You can then resize the window and move it away from the central work area. Scroll through the text as you work through the model.

Model Parameters

The tables in this section describe the various parameters and list their default values.

Drain-Source Current Parameters

The drain-source current model was developed through examination of Ids, as well as drain-source conductance (Gds) and transconductance (Gm), vs. bias plots on a wide class of devices from various manufacturers. The model assumes the device is symmetrical, and therefore is valid for values of Vds < 0.0 V as well as Vds > 0.0 V. The parameters in the following table are extracted from drain-source current measurements:

Table 36   Drain-Source Current Parameters

Parameter	Description	Default (Units)
Vto	Zero-bias threshold parameter.	−1.5 (V)
Gamma	Vds-dependent threshold parameter.	0.05 (1/V)

Table 36   Drain-Source Current Parameters (continued)

Parameter	Description	Default (Units)
Vgo	Gate-source voltage where transconductance is maximum.	−0.5 (V)
Vdelt	Parameter that controls linearization point for Gm characteristic.	0.0 (V)
Vch	(Vchannel) Gate-source voltage where Gamma no longer affects the I-V curve.	1.0 (V)
Gmmax	Peak transconductance parameter.	70.0e−3 (S)
Vdso	Drain voltage where Vds dependency disappears from equations.	3.0 (V)
Vsat	Drain-source current saturation parameter.	1.0 (V)
Kappa	Output conductance parameter.	0.0 (1/V)
Peff	Channel-to-backside self-heating parameter (effective power compression).	2.0 (W)
Vtso	Subthreshold onset voltage.	−10.0 (V)

Charge Parameters

The gate charge model was developed through examination of extracted device capacitances over bias. The capacitance data is obtained from measured Y-parameter data in the saturation region.

Table 37   Charge Model Parameters

Parameter	Description	Default (Units)
C11o	Maximum input capacitance for Vds=Vdso and Vdso>Deltds.	0.3 (pF)
C11th	Minimum (threshold) input capacitance for Vds=Vdso.	0.03 (pF)

Table 37   Charge Model Parameters (continued)

Parameter	Description	Default (Units)
Vinfl	Inflection point in C11–Vgs characteristic.	−1.0 (V)
Deltgs	C11th to C11o transition voltage.	0.5 (V)
Deltds	Linear region to saturation region transition parameter.	1.0 (V)
Lambda	C11–Vds characteristic slope parameter.	0.05 (1/V)
C12sat	Input transcapacitance for Vgs=Vinfl and Vds>Deltds.	0.03 (pF)
Cgdsat	Gate-drain capacitance for Vds>Deltds.	0.05 (pF)
Ris	Source-end channel resistance.	2.0 (ohms)
Rid	Drain-end channel resistance.	0.0 (ohms)
Tau	Source-to-drain charging delay.	1.0 (ps)
Cdso	Drain-source inter-electrode capacitance.	80.0 (fF)

Dispersion Parameters

Dispersion parameters are extracted from measured S-parameter data, to provide additional accuracy for dispersion effects at higher frequencies. The dispersion parameters listed in the table below are optimized to measured transconductance (Gm) data to model both DC and RF dispersion characteristics. Similarly, the dispersion parameters are optimized to measured output conductance (Gds) data.

Table 38   Dispersion Model Parameters

Parameter	Description	Default (Units)
Rdb	Dispersion source output impedance.	1.0e9 (ohms)
Cbs	Dispersion source capacitance.	1.6e−13 (F)

Table 38   Dispersion Model Parameters (continued)

Parameter	Description	Default (Units)
Gdbm	Additional d-b branch conductance at Vds=Vdsm.	0.0 (S)
Kdb	Parameter that controls Vds dependence of additional d-b branch conductance.	0.0 (dimensionless)
Vdsm	Voltage where additional d-b branch conductance becomes constant (Gdbm).	1.0 (V)
Gmmaxac	Peak transconductance parameter (AC).	60.0e−3 (S)
Vdeltac	Parameter that controls linearization point for Gm characteristic (AC).	0.0 (V)
Vtoac	Zero-bias threshold parameter (AC).	−1.5 (V)
Gammaac	Vds-dependent threshold parameter (AC).	0.05 (1/V)
Kappaac	Output conductance parameter (AC).	0.0 (1/V)
Peffac	Channel-to-backside self-heating parameter (AC).	10.0 (W)

HEMT Gm Compression Parameters

Some FETs and almost all HEMTs exhibit transconductance compression in which Gm peaks and then decreases. The following parameters model these compression effects.

Table 39   Additional Parameters Used in EEHEMT1 for Gm Compression

Parameter	Description	Default (Units)
Vco	Voltage where transconductance compression begins.	10.0 (V)
Mu	Parameter that adds Vds dependence to transconductance compression onset.	0.0 (dimensionless)

Table 39   Additional Parameters Used in EEHEMT1 for Gm Compression

Parameter	Description	Default (Units)
Vba	Transconductance compression "tail-off" parameter.	1.0 (V)
Vbc	Transconductance roll-off to tail-off transition voltage.	1.0 (V)
Deltgm	Slope of transconductance compression characteristic.	0.0 (S/V)
Alpha	Transconductance saturation-to-compression transition parameter.	1.0e−3 (V)

Calibrating the Network Analyzer

All the EEFET3/EEHEMT1 S-parameter measurements require only one calibration: a swept broadband calibration across the operating frequency range of the device under test. Good calibration of the network analyzer is critical to a good measurement and extraction.

For step-by-step calibration procedures, refer to:

• Agilent 8510-based system—"Calibrating the Network Analyzer" on page 589 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

• Agilent 8753-based system—"Calibrating the Network Analyzer" on page 631 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

• Wiltron 360 network analyzer—Refer to the Wiltron documentation

Make the following additional setting changes:

• Set the frequency range for the calibration to the range you intend to use for your measurements, at least as wide as the operating range of the device.

- Set the network analyzer START frequency to the lowest possible frequency in the range of the network analyzer and bias networks: low frequencies allow more accurate AC conductances to be extracted. The example procedure in the software uses 50 MHz.

- Set the STOP frequency to the upper limit of the frequency range for the device under test. Frequencies up to Ft are useful but not required. Good package and charge extractions require frequencies high enough to see roll-off and package effects clearly. The example procedure uses 3 GHz.

- Set the NUMBER of POINTS to 51 or 101 to provide clear and usable S-parameter plots: more points only add more time and memory requirements to the measurement and extraction.

Remember these frequency values: you will also need to enter them in IC-CAP when instructed. If you do not set these values correspondingly, the calibration will be invalid.

All S-parameter measurements for the EEFET3/EEHEMT1 model are performed over the same frequency range.

NOTE	If you set a MHz value in IC-CAP, write *MEG* in full. IC-CAP is not case-sensitive and assumes that *m* or *M* corresponds to *milli.* If an error message is displayed during the measurement that indicates a network analyzer frequency value is too low, you may have entered *M* only.

Remember the number of the network analyzer cal set or register where you store the calibration. You will need to use this number in the instrument options of *each* S-parameter measurement setup when you define the instrument states.

Using Macros for Measurements and Extractions

Macros automate the modeling procedure so that a series of similar devices can be modeled quickly with minimal user intervention. As a macro is run, it sets the control and device measurement variables in the model variable table usually

needed for the extraction procedure being run. The control variables are explained in the next pages. These are the macros and their purposes:

Table 40   Macro Descriptions

Macro	Description
Set_Measurement_Ranges	Execute the device preview (including Yang_Long_Preview) and initialization steps, setting all the control variables.
Package_extraction_only	Measure the data from which the package parasitics are extracted, and perform the extractions.
DC_extraction_only	Makes the DC measurements and performs the DC extractions.
AC_extraction_only	Makes the S-parameter measurements and performs the AC extractions.
Measure_all_data	Executes all measurements
Extract_all_parameters	Extracts model parameters based on existing data
Measure_and_Extract_ALL	Performs a complete device measurement and extraction using the measurement ranges recorded in the model variable table
Xtract	Performs the parameter extraction procedure. Called by all the other macros except Set_Measurement_Ranges.

Macro Sequence

One possible appropriate sequence for performing the macros is as follows:

1 Set_Measurement_Ranges

2 Package_extraction_only

3 DC_extraction_only (and study the results)

4 AC_extraction_only (and study the results)

Control Variable Flags in the Macros

Control variable flags are set to 1 or 0 in the macros to simplify operation and customize the macros. Each macro starts with default values for the control variables, so they can be used as is or modified to meet your needs. For details, refer to "Running the Macros" on page 313.

Table 41 Macro Control Variables

Control Variable	Set to 1	Set to 0
control_view	Displays the plots.	Suppresses the plots.
control_measure	Performs the measurements.	Skips the measurement steps. Used with previously-stored data.
control_extract	Executes the parameter extraction.	Skips the parameter extraction.
control_package	Executes the package extraction.	Skips the package extraction.
control_dc_model	Executes the DC measurement and extraction.	Skips the DC procedures.
control_ac_model	Executes the AC measurement and extraction.	Skips the AC procedures.
control_save	Saves a Series IV compatible file.	Does not save a Series IV compatible file.

The control variables can also be set in the variable table, but control variable values set in the macros override variable table values.

To see more detail on the macro flags, you can select any macro to read the contents of the macro.

Running the Macros

1 In the Model window, select the macro of your choice.

2 To change control variable flags (see Table 42), use Back Space or Delete and replace the current value with a different value.

NOTE	If you want to see the control variable values change when you execute the macro, Detach the model variable table.

3 To start the macro, click **Execute**. The macro will start to run. You can see the control variable values change in the model variable table if you have modified them in the macro.

4 If you select the Set_Measurement macro, you will be prompted to enter values for certain variables. Otherwise the variable table default values are used. Note that the variable values must be appropriate to the particular device being modeled, or the macro results will not be valid.

NOTE	You will need to know several things about the device under test to supply appropriate values. For example, you will be prompted for the device gate width and length. You also need to supply certain voltage or current values. Some of these values can be found in the device manufacturer's data sheet. The list of device input variable values later in this chapter also lends insight into appropriate values for different variables. Default values are also available, and may be appropriate for use in many cases.

The macro will set numerical values for the control and device measurement variables (see Table 42) needed for the measurement and extraction procedure being run. Then the macro will perform the required measurement and extraction procedures.

If you have the control_view flag set to 1, you will see the measured and extracted data plotted on the screen.

When the macro is complete, the results of the extraction are displayed in the Status window.

Device Input Variable Values

One of the first requirements in performing the EEFET3/EEHEMT1 modeling procedures is to set numerical values for the model variables. When executing the macros, you are prompted for several variable values. Other variables are calculated by the software based on your input. The Set_Measurement_Ranges macro is especially useful in establishing initial settings in preparation for a complete extraction. We recommend that you use this macro for initialization of the model variable table.

Model Variables

The EEFET3/EEHEMT1 model variables are described in the following table.

Table 42   EEFET3/EEHEMT1 Model Variables

Variable	Description
CAP_FACTOR	Used as a scaling factor in setting charge parameter optimzation limits.
CHECK_DATA	Used in the macros to control extent of application/user interface. Typically opens plots and performs transforms for visual inspections.
control_ac_model	Used in the macros to control the AC measurement and extraction procedures. 1 performs the procedures, 0 does not.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
control_dc_model	Used in the macros to control the DC measurement and extraction procedures. 1 performs the procedures, 0 does not.
control_doc	Not used in this Model.
control_expert	Used to control the level of interaction between the application and the user. 1 non verbose, 0 verbose.
control_extract	Used in the macros to control the parameter extraction steps. 1 executes the extraction, 0 skips the extraction.
control_frequency	Used in Set_Measurement Macro to set input responses which guide the user in determining whether or not to initialize Frequency settings and Instrument Options for Network Analyzer (NWA). 1 to guides the user through NWA settings, 0 does not.
control_hemt	Determines whether the device will be modeled as a FET or a HEMT. It is normally set to 1 for a HEMT, and 0 for a FET. However, the HEMT setting generally works well with standard MESFETs also, providing a better Gm curve fit. In particular, it is suggested you use the HEMT setting for a device where the Gm curve peaks and then rolls off as Vgs becomes more positive.
control_measure	Used in the macros to control the measurement steps. 1 performs the measurements, 0 suppresses the measurements.
control_package	Used in the macros to control the package parasitics measurement and extraction procedures. 1 performs the procedures, 0 does not.
control_save	Used in the macros to save a Series IV compatible file. 1 saves the file, 0 does not.
control_view	Used in the macros to control display of the plots. 1 displays the plots, 0 suppresses the plots.
control_YL_method	Used in Xtract macro to guide the extraction process. Also used in the method for selecting Rs. 1 selects Yang Long method, 0 does not.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
FC	The frequency at which the charge model is extracted. Use a frequency value that gives about 90 degrees of rotation of the S-parameters measured.
FG	The frequency at which device conductances are extracted. Experience has shown that data just above the trap effects is the best for extraction of conductances.
FREQ_POINTS	The number of points for S-parameter measurements. This must be the same as the number of points set for the network analyzer calibration. 51 or 101 provides clear and usable S-parameter plots: more points only add more time and memory requirement to the measurement and extraction.
FREQ_START	The start frequency of the network analyzer for S-parameter measurements. Use the lowest possible frequency in the range of the network analyzer and bias networks: low frequencies allow more accurate AC conductances to be extracted. This must be the same as the start frequency set for the network analyzer calibration. (See "Calibrating the Network Analyzer," later in this chapter.)
FREQ_STOP	The stop frequency of the network analyzer for S-parameter measurements. This should be the upper limit of the frequency range for the device under test, and must be the same as the stop frequency set for the network analyzer calibration. Frequencies up to Ft are useful but not required. Good package and charge extractions require frequencies high enough to see rolloff and package effects clearly.
FS	The frequency at which Tau and Ri are extracted. It must be about where S11 crosses the real axis: this is the point where the impact of the device parasitics causes the least problem.
I_RES	The current resolution of the power supply being used. This value is used in macros to help truncate current limits to values that do not violate the resolution limits of the DC power supplies in use.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
Id_per_Wg_factor	An empirical based scaling factor used for preliminary determination of Wg (gate Width) or IDSS, Set_Measurment Macro.
ID0_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
I*D1_MAX_SUPPLY*	Used in determining if Power Supply limits have been or will be exceeded.
ID2_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
ID3_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
IDS_FACTOR	Use as a scaling factor for setting the optimization range of DC-IV equation parameters: gmmax, vgo, vto etc.
IDS_MAX	The compliance limit for Ids values. (The Agilent 41420A high-power SMU likely to be connected to the device drain has a maximum output current of 1.0A. A standard Agilent 11612A option K10/K20 bias network has a maximum input current of 0.5A.)
IDS_NOM	The intended Ids of device operation.
IDS_VTO_FACTOR	The fraction of the maximum drain current that will define the drain current at Vto.
IDS_VTO_MAX	Calculated value of Ids a Vto. (2 * *IDS_VTO_MAX*)
IDS_VTO_MIN	Calculated value of Ids at Vto.
IDS_YL_MAX	The upper limit of Ids used to extract Rs. It also will be different for every transistor type and size. You will select this value from the data plot presented by the Yang_Long_Preview measurement. The value must be within 10% of *IDS_YL_MIN*.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
IDS_YL_MIN	The lower limit of Ids used to extract Rs. It will be different for every transistor type and size. You will select this value from the data plot presented by the Yang_Long_Preview measurement. The value must be within 10% of *IDS_YL_MAX*. Remember to use values within the settability limits of the DC source/monitor.
IDS_YL_PTS	The Yang_Long method requires swept measurements of two slightly different values of drain current, therefore the standard number of Ids points is 2.
IDSS_ACTUAL	Used to store value of Idss from measured data. Also used in calculations of other variables. In some cases IDSS_ACTUAL is assigned the value *IDSS_DATA_SHT*.
IDSS_DATA_SHT	The Idss value for the device. This may be specified in a manufacturer's data sheet. If you do not know an expected value for Idss, you can use the default value.
Ig_density	The gate current density of the device, typically 50×10^6 A/m^2. The software uses the gate width and length values, in combination with the gate current density, to calculate safe forward gate bias levels.
IG_FWD_IV	This is the maximum allowable DC gate current. This value is used to measure *VG_FWD_IV*, which in turn is used to set the value of VGS_MAX, the highest Vgs used in swept bias measurements.
IG_FWD_MAX	This is the gate current at *VG_FWD_MAX*, and should be slightly less than the critical gate current IFwd but high enough to reduce the gate diode's dynamic resistance to a few ohms. It is used in the ig_vgs device preview setup.
IG_FWD_MID	This is a current about 95% of the value of *IG_FWD_MAX*. It is used to measure *VG_FWD_MID*.
IG_FWD_MIN	This is a current about 95% of the value of *IG_FWD_MID*. It is used to measure VG_FWD_MIN.
IG_FWD_RATIO	The percent reduction of Ig from *IG_FWD_MAX* to IG_FWD_MID to IG_FWD_MIN. It is usually approximately 92% to 98% as specified by Dambrine and Cappy for their method of extracting Rg and Rd.

Table 42 EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
IG_MAX	Not used in this model.
IGS_MAX	The compliance limit for Igs values. The default value, 0.100A, is the maximum output current of the Agilent 41421B medium-power SMU likely to be connected to the device gate. You provide compliance values to limit SMU output current or voltage and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used.
IGS_YL_MAX	Select this value in the linear region of the Yang_Long_Preview plot, at the upper end of the region, and where the Igs value is between 50 and 100 times smaller than the maximum Ids value selected.
IGS_YL_MIN	For the Rs extraction Vgs is measured between *IGS_YL_MIN* and *_MAX*. Like the Ids values, you will select the Igs values from the data plot presented by the Yang_Long_Preview measurement. Select an *IGS_YL_MIN* value in the linear region of the plot, near the lower end, and where the Igs value is between 50 and 100 times smaller than the minimum Ids value selected.
IGS_YL_PTS	The number of Igs points used in the Rs extraction measurement. Remember in setting these last three Igs values to use limits and number of points values that result in steps within the settability limits of the DC source/monitor: the steps will be within uA spacing.
Lg	The gate length of the device, in microns.
MEASURE_FAST	Not used in this model.
model_file_name	This is used to define the disk file name.
model_name	Current name of model. Used in macro to specify iccap path (example: model_name > DUT > Setup > Plot)
Pdiss_MAX	Maximum safe power dissipation of the device. This is used to check the VDS_MAX value once Idss is known.
RANGE_FACTOR	Used to adjusts the limits of optimization for the dynamic IV parameter set.
Rch_factor	Not used in this model.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
SIMULATOR	not used in this model.
TOL_VOLT	Not used in this model.
TOL_FUNC	Not used in this model.
TRACE_MAX	This sets an upper limit for the optimization range of Vds for the gm_gds optimization in the AC_all DUT.
TRACE_MIN	This sets a lower limit for the optimization range of Vds for the gm_gds optimization in the AC_all DUT.
V_RES	The voltage resolution of the power supply being used. This value is used in macros to help truncate voltage limits to values that do not violate the resolution limits of the DC power supplies in use.
V_RES_vto	Used to set the correct resolution for Vgs.
VD1_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
VD2_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
VD3_MAX_SUPPLY	Used in determining if Power Supply limits have been or will be exceeded.
VDELT_FACTOR	Used as a scaling factor when scaling the parameter EEFET3.vdelt.
VDS_COARSE	The number of Vds points used for making DC-biased S-parameter measurements. A number between 5 and 15 results in a good extraction.
VDS_FINE	The number of Vds points used in gate characterization steps. 51 points is an adequate number.
VDS_MAX	The largest safe Vds value for all Vgs values. Be aware of the power dissipation limits of your device at high Ids levels, to prevent damage to the device.
VDS_MIN	The smallest value of Vds. This is usually 0.0V, but may be a negative value.
VDS_NOM	The intended Vds of device operation.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
VDS_YL_LMT	The maximum allowable Vds for Rs measurement and extraction.
VDS_YL_MAX	The Rs extraction must be done at Vds values less than 0.25V. This guarantees the extraction takes place in the region of linear transistor operation. Use a value at the upper end of the linear region. These Vds values generally will never have to be changed except in the case of a device with a very sharp knee.
VDS_YL_MIN	This is the lower limit of the Vds range for the Rs extraction, which must be done at Vds values less than 0.25V. Use a value in the linear region, near the low end of the region.
VDSO	A Vds value just above the knee in full saturation. This is the point where the model equations simplify, and is an important operating characteristic for most GaAs FETs. Manufacturers often specify Idss at values of Vds that generally work well as Vdso settings. Vdso=3.0V is a good default that will nearly always work.
VG_FWD_IV	A measured Vgs value where a normally small forward quiescent DC gate current flows for an active bias condition. It is used to help determine *VGS_MAX* for DC and AC measurements.
VG_FWD_MAX	The upper limit of Vgs at *VDS_ZERO* for the rg_rd cold fet measurement.
VG_FWD_MID	VG_FWD_MID is the gate voltage used to produce IG_FWD_MID.
VG_FWD_MIN	This is the lower limit of Vgs with Vds at *VDS_ZERO*, used to measure S-parameters at forward gate bias in the parasitic resistance measurement.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
VG_FWD_PTS	The number of points for the rg_rd measurement. This is the number of measurements you wish to make between *VG_FWD_MIN* and *VG_FWD_MAX*, inclusively. Three points are sufficient.
	This following series of _YL_ variables are all used in the Yang_Long measurements from which the parasitic source resistance Rs is extracted. It is highly recommended that the initialize transform be used to set these values once the Yang_Long_Preview measurement is made. The preview measurement is based on voltages that are nearly universally applicable. The initialize transform extracts the highly device-dependent currents needed to use this method of extracting RS. Refer to the procedures later in the chapter.
VG_GML_MAX	The largest Vgs value used for Gummel measurements. Since this data is not used to extract contact resistances, use values that yield Igs values $\leq$ IFwd/2.

Note Remember that the Agilent 4142 DC source/monitor and its SMUs are limited in the ability to set and resolve voltage steps. In all cases where a measurement will be stepped over a number of points, set values that will make the voltage step size equal to an integer multiple of the Agilent 4142 voltage resolution. Voltage steps in fractional mV settings are not valid and will not work. The "Getting Started" chapter of the Agilent 4142B Operation Manual has more information on the voltage resolution of Agilent 4142B SMUs.

Variable	Description
VG_GML_MIN	The smallest gate characterization voltage, used for Gummel measurements. Data taken over this range is used in the gate diode and source resistance extractions. Values lower than about 0.2V to 0.3V result in Igs data that is indistinguishable from noise.
VG_GML_RANGE	A minimum difference allowed between *VG_GML_MAX* and *VG_GML_MIN*, usually 0.5V. It helps to keep the gate characterization measurements within a linear, noise-free range.
VGO_FACTOR	Used to initialize the parameter EEFET3.vgo.

Table 42   EEFET3/EEHEMT1 Model Variables (continued)

Variable	Description
VGS_COARSE	The number of Vgs points used for making DC-biased S-parameter measurements. S-parameters will be measured at each of these bias points at every calibrated frequency point. If the VGS_COARSE number of points is specified too large, more data than necessary will be taken and the measurement will take an excessive amount of time. A number between 5 and 15 results in a good extraction.
VGS_FINE	The number of Vgs points used in gate characterization steps. 101 points is an adequate number. Remember that voltage steps in fractional mV increments do not work.
VGS_MAX	The highest (most positive) Vgs value where the device can be operated quiescently at VDS_MAX.
VGS_MIN	A Vgs value that gives even, easy-to-use steps and is more negative than the device pinchoff voltage at VDS_NOM.
VGS_NOM	The value of Vgs that gives an Ids value representative of actual device operation.
VGS_YL_MAX	The Vgs compliance for the Yang_Long_Method. This protects the device from overdrive on the gate. A value of 0.6V will work for nearly all devices used, though occasionally it may need to be set to 0.7V.
VTO_FACTOR	Not used in this model.
Wg	The gate width of the device, in microns.
YL_IdIg_RATIO	The factor used to compute acceptable gate currents as a function of drain current in Yang and Long's source resistance method. Usually between 50 and 100.

Executing Individual Measurement Transforms

While the macros are appropriate for measuring and modeling devices for which the device type is well known, you may prefer to set the variable values yourself rather than have them set by the macros. Or situations may occur where the device is not

known and you need to perform the device measurements separately and extract the parameters individually from the measured data.

When executing individual transforms, you will sometimes be prompted to supply variable values. You will also input some variable values directly in the model variable table.

The rest of this chapter explains in detail how to:

- Make preview device measurements and set the model variables from the results obtained

- Make the measurements required for specific extractions

- Perform the individual extractions

- Save the resulting model

The Preview DUT

The Preview DUT contains three setups, as shown in the following illustration:
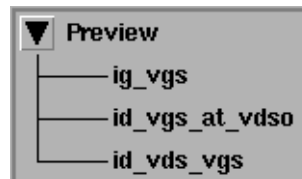
These setups are used to obtain a good set of measured DC-IV curves on which most of the other measurements will be based. In performing these measurements, you set the variable values to define the measurement limits. Some values are selected to protect a device from damage. Other values cause data to be taken over a wide enough range to ensure usable model results, or taken in specific regions of interest to you.

Begin with anticipated reasonable values depending on your knowledge of the device under test. Then modify the values as necessary until you are satisfied with the measurement results.

The validity of the later measurements and the accuracy of the model are dependent on obtaining a good set of preview measurements. Note, however, that when extracting many devices of the same type, repeated device preview is not necessary.

NOTE	The Set_Measurement_Ranges macro performs all the steps outlined here. A thorough understanding of what is done in these steps and how it fits into the extraction methodology will make the extraction process go more smoothly.

Parameter extraction must determine the bounds of a transistor's operation without distorting or destroying the part. You need some knowledge of the device's intended regions of operation to perform the device preview steps. The most accurate model performance is achieved when the intended use of the model is accounted for during measurement and extraction.

The EEFET3/EEHEMT1 model does not test the breakdown characteristic of the GaAs FET. That value can be obtained from the results of destructive testing. The value of breakdown voltage is used by the model to establish the effects of large Vds values. It is assumed this value is known.

The device preview measurement procedures following provide explanations of setting the new values, and how the screen prompts, the variable table, and the input tables interact.

NOTE	When you input variable values in the software, in macros or in non-automated procedures, they must be in numerical form. If you input non-numeric text when IC-CAP is expecting a number, IC-CAP will interpret it with a value of 0.0.

Preview / ig_vgs

This setup measures gate and drain voltage as a function of gate current. Four specific values of forward gate current are applied, with the drain current set to a constant value of 0.0A.

The purpose of the setup is to use the gate current density of the device, and the gate width and length, to set the proper variable values for gate voltage.

This setup determines several gate voltages that will be used in later measurements, and sets their values in the variable table. These variables are VG_FWD_MAX, VG_FWD_MID, VG_FWD_MIN, and VG_FWD_IV.

The input variables you set in the following procedure will configure the FET as shown in the following figure.
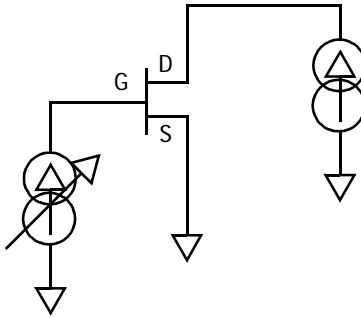
Figure 109   FET Bias Configuration for ig_vgs Measurement

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Initializing The Input Variable Values

The inputs control the measurement. The variables listed in the inputs use the numerical values from the variable table. You define the actual numerical values in response to prompts from the software. You generally do not need to change the inputs, but they are illustrated and explained for your information.

The values you use will be dependent on your knowledge of your device under test. As you observe the results of the preview measurements, you can change the variable table values until you are satisfied with the measured results.

1 If you have not already connected the device, do so now. Observe antistatic precautions.

NOTE   Remove any high-intensity light sources such as microscope light before taking a measurement. One easy method of blocking light is to place an opaque plastic box on the fixture directly above the device.

2 Select **Preview > ig_vgs**.

Notice the inputs ig and id. These are used by the software to control the measurement. Note that Mode is set to I to provide a current to both ig and id inputs.

```
      Input: ig                    Input: id
      Mode: I                      Mode: I
   To Node: G                   To Node: D
 From Node: GROUND            From Node: GROUND
      Unit: SMU1                   Unit: SMU2
Compliance: VGS_YL_MAX      Compliance: VDS_MAX
Sweep Type: LIST            Sweep Type: CON
Sweep Order: 1                   Value:  0.000
# of Values: 4
   Value 1: IG_FWD_MAX
   Value 2: IG_FWD_MID
   Value 3: IG_FWD_MIN
   Value 4: IG_FWD_IV
```
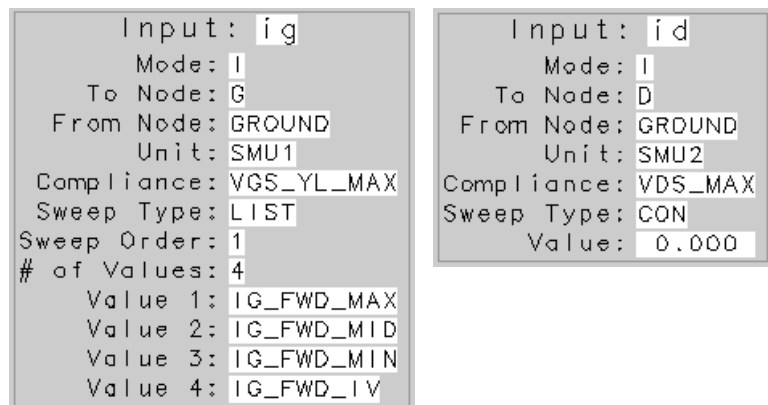
Figure 110   Preview / ig_vgs inputs

The Unit names must conform to the names you set in the hardware window (see "Getting Started in IC-CAP" on page 300). They must also correspond with the actual plug-in SMU connections from the DC source/monitor. SMU1 is the gate supply and SMU2 is the drain supply. Since the source is grounded, only the gate and drain inputs need to be set.

Compliance values limit SMU output voltage or current and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used. With a current input, compliance refers to voltage. In the model variable table, set the Value of VGS_YL_MAX to a low value that will protect the device from overdrive on the gate. A value of 0.6V will work for nearly all devices, though occasionally it may need to be set to 0.7V or higher. Be sure to use a value within the resolution settability limits of the DC source/monitor.

3 In the model variable table, set the value of VDS_MAX to the largest safe Vds value for all Vgs values. Be aware of the power dissipation limits of your device at high Ids levels, to prevent damage to the device.

The Sweep Type for ig is set to List to allow four specific different gate current inputs. In list sweep mode, it is possible to specify up to 15 different bias inputs for the measurement. This measurement requires inputs of IG_FWD_MAX, IG_FWD_MID, IG_FWD_MIN, and IG_FWD_IV.

4 In the variable table, locate the values for IG_FWD_MAX, IG_FWD_MIN, and IG_FWD_IV.

5 From the Extract/Optimize folder, select the **update_ranges** transform and click **Execute**. The transform will check for data and then update the appropriate variables.

6 The updated variable values will be listed in the Status window.

Preview / id_vgs_at_vdso

This setup measures drain current as a function of swept gate voltage, with drain voltage set to a constant bias value equal to Vdso. The gate voltage is swept across the operating range from

minimum to maximum. The measurement covers all regions of device operation from subthreshold through threshold and high current regions.

Vdso is an important operating characteristic for most GaAs FETs. The model's IV equations simplify greatly at Vdso. Setting Vdso appropriately helps to ensure that the rest of the IV and the dispersion parameters will extract efficiently.

This setup determines several gate and drain voltage and current variables, and sets their values in the variable table.

The input variables you set in the following procedure configure the FET as shown in the following figure.
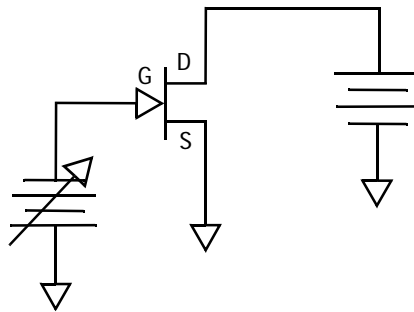
Figure 111   FET Bias Configuration for id_vgs_at_vdso Measurement

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

NOTE

Use the same settings as you did for the ig_vgs measurement, except set Integ Time to L (long) for maximum averaging for this measurement.

Then return to this chapter and continue.

Initializing the Input Variable Values

1 Select **Preview > id_vgs_at_vdso**.

   Notice the inputs vg and vd (see the following figure). Note that Mode is set to V to provide a voltage to both vg and vd inputs. With a voltage input, compliance refers to current.

```
   Input:  vg                      Input:  vd
      Mode: V                         Mode: V
    + Node: G                       + Node: D
    - Node: GROUND                  - Node: GROUND
      Unit: SMU1                      Unit: SMU2
Compliance: IGS_MAX             Compliance: IDS_MAX
Sweep Type: LIN                Sweep Type: CON
Sweep Order: 1                       Value: VDSO
      Start: VGS_MIN
      Stop: VGS_MAX
# of Points: VGS_FINE
  Step Size:
```
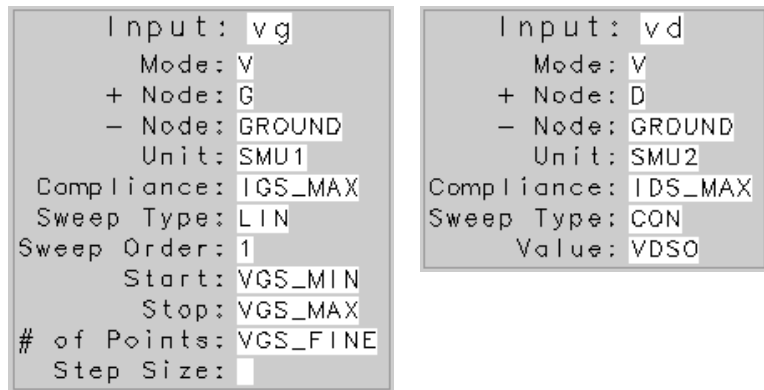
Figure 112   Preview / id_vgs_at_vdso inputs

2 In the model variable table, set the Value of IGS_MAX no greater than the maximum allowable gate current for your device in amps. Be sure to use a value within the resolution settability limits of the DC source/monitor. Also note that the maximum output current of the Agilent 41421B medium-power SMU likely to be connected to the device gate is 0.100 amp.

3 In the model variable table, set the Value of IDS_MAX lower than the maximum current for your device. The maximum output current of the Agilent 41420A high-power SMU likely to be connected to the device drain is 1.0 amp.

- In this model, the compliance for vg is always IGS_MAX and the compliance for vd is always IDS_MAX.

- The Sweep Type for vg is set to LIN to provide a linear voltage sweep from start to stop gate voltage values. The Sweep Type for vd is set to CON for a constant drain voltage value equal to Vdso.

4 Set VGS_MAX to the highest (most positive) Vgs value where the device can be operated quiescently at VDS_MAX. (VDS_MAX is the largest safe Vds value for all Vgs values.)

5 Set VGS_FINE to a fairly large number of points, sufficient to display the detail of the measurement across the trace. Typically, 101 points is an adequate number. Note that voltage steps in fractional mV increments do not work.

6 From the Extract/Optimize folder, select the **initialize** transform and click **Execute**. The plots will be displayed, and screen prompts will ask you to input values for other variables needed in this measurement. Use values appropriate for your device.

VDSO is one of the most important operating characteristics set in the model. Usually a transistor's static IV curves will *peak up* a little just after the device enters the saturation region. The value of Vds at this peak is a perfect value for Vdso. If the device shows no *hump* in the IV curves at about Vds=3.0V, select a Vdso value approximately 1V above the knee of the Vgs=0 curve. Vdso=3.0V is a good default and will nearly always work. It is important that Ids is at its quiescently measured maximum at this point. Device manufacturers often specify Idss at a value of Vds that is ideal as a setting for Vdso.

When you have responded to all the prompts, the transform measures the device and updates the plots. It sets appropriate numerical values for the relevant variables, and lists them in the model variable table and the Status window. Preliminary values for some of the model parameters will also be listed.
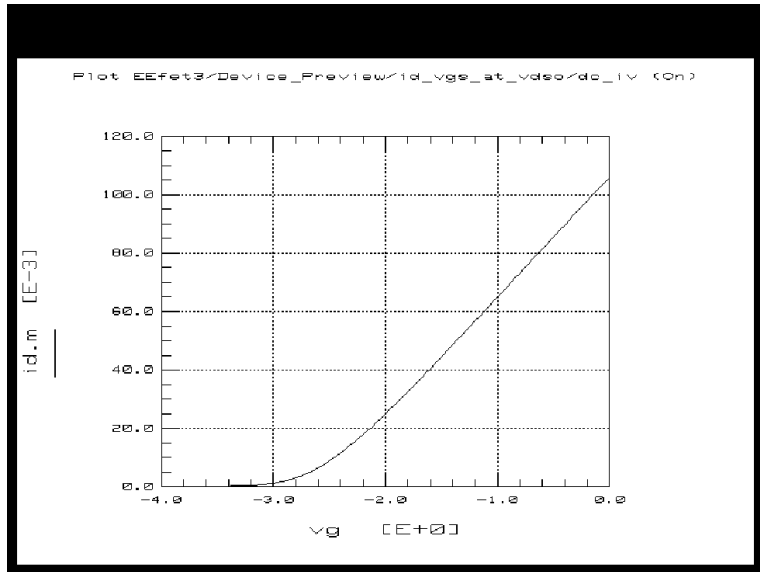
Figure 113   Example dc_iv Plot in the id_vgs_at_vdso Setup

The dc_iv plotted data should resemble the previous figure in shape, though the values will be specific to your device. It is most important to include data points in the subthreshold region. Note that as Vds becomes more positive the device threshold will tend to become more negative. If the data indicates the device is not operating in the correct range, you will need to redo the initialization and change the input variable values you set, particularly the VGS_MIN and/or VGS_MAX values.

- If the measured data looks wrong for your device, check the probe contacts, the bias connections, and the measuring instruments. Make sure the device is oriented properly, with the right ports connected to the right terminals. If none of these resolves the problem, the device may be defective.

- If you want to save the measured data for later comparison with data predicted by the extracted model, choose **File > Save As**. Enter an appropriate filename (for example *id_vgs_at_vdso*), and click **OK**.

Preview / id_vds_vgs

This setup measures drain current with respect to drain voltage, at several values of gate voltage. The drain voltage is swept from 0V to the upper limit of its operating range. The measurement covers the overall region of device operation, and plots the characteristic IV curves vs. Vds.

The input variable values you set will configure the FET as shown in the following figure.
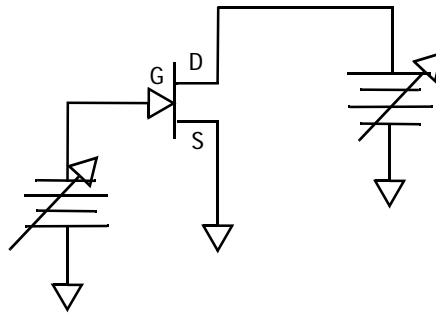
Figure 114   FET Bias Configuration for id_vds_vgs Measurement

Defining the DC Source/Monitor Instrument State

The instrument states need to be set independently for each setup. Use the same settings as you did for the id_vgs_at_vdso setup, *except* that you can set Integ Time to M (medium) to allow a faster measurement time: this measurement does not require maximum averaging.

Follow the steps described in the appropriate chapter shown next:

• "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

• "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Then return to this chapter and continue.

Initializing the Input Variable Values

1 Select **Preview > id_vds_vgs** setup and observe the inputs vg and vd (see the following figure).
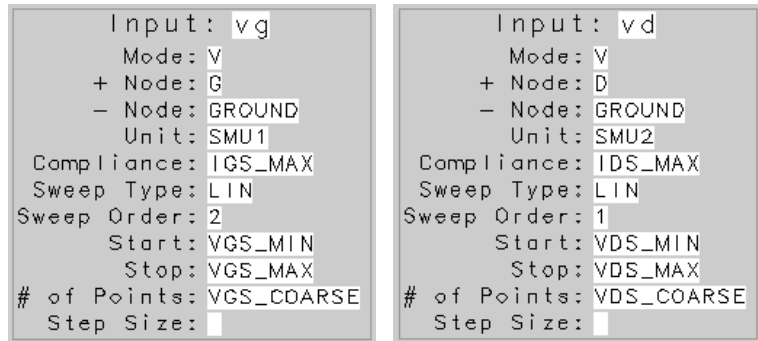
Figure 115 Inputs for id_vds_vgs Measurement

- Notice that for this measurement, both vg and vd are set with a Sweep Type of LIN to provide a linear sweep from minimum to maximum values.
- Note that the Sweep Order for vd is set to 1, and for vg is set to 2.
- The Start and Stop values for vg are set to VGS_MIN and VGS_MAX as in the previous measurement. However, # of Points is now set to VGS_COARSE. This represents the number of separate measurement traces that will be made of Id at evenly-stepped Vgs values. In the variable table, set the VGS_COARSE value to a fairly low number, between 5 and 15, so that the separate traces will be distinguishable. Remember that fractional mV step values will not work.

2 For VDS_COARSE, set a number sufficient to display the measurement clearly across the trace but without the need for detail: an arbitrarily large number of data points does not increase model accuracy. A VDS_COARSE value between 5 and 15 will provide enough detail without unduly slowing the measurement.

3 From the Plots folder, select the **dc_iv** plot and click **Display Plot**. A plot of the device characteristic curves will be displayed.

4 Set VDS_MIN to the smallest value you will use for Vds. This is usually 0.0V, but may be a negative value.

5 Set VDS_MAX to a value in saturation well above Vdso. This should be the largest safe Vds value for all Vgs values. Be aware of the power dissipation limits of your device at high Ids levels, to prevent damage to the device: Pdiss_MAX must not exceed VDS_MAX*IDS_MAX.

When you have input all the necessary values, click **Measure**. This measures the device and updates the plot. Review the plot and if needed change the necessary variables and re-measure.
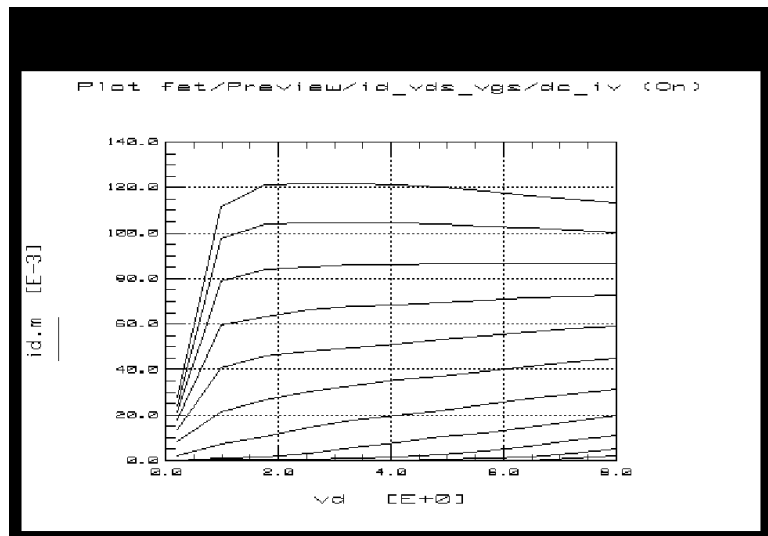
Figure 116   Example Measured ids_vds_vgs Device Preview Data

The plotted data should resemble the previous figure. If it does not, you may need to change the VDS and/or VGS variables. Continue until you have an appropriately nominal set of characteristic IV curves displayed, showing good coverage of the Vds and Vgs features.

6 Optionally, to save the measured data, choose **File > Save As**. Enter an appropriate filename and press Enter or click **OK**.

7 Close the plot.

The Source_Resistance DUT

There are many different approaches to the measurement and extraction of the common lead resistance in junction FET devices. Perhaps the most common is the "end resistance" measurement method where the unbiased drain terminal is used as a voltage probe to measure the voltage at the top end of Rs while a substantial current is forced into the gate. While that method is fairly good, it includes inaccuracy due to channel resistance effects. The method presented here overcomes the problem by correcting for the effect of the open channel resistance (see "End Resistance Technique" on page 398).

Modified End Resistance Measurement Technique

The Yang and Long method computes the value of Rs from the change in Vgs due to the application of two drain currents, which are 50 to 100 times greater than Ig. The two drain currents are chosen to be within the drain's linear IV region (Vds < 0.25 volts).

The data for the computation of Rs is taken by sweeping Ig over a range that results in measured Vgs values in the range of 0.300V to 0.600V. Two separate Ig sweeps are made, with current forced at the drain terminal at two levels. The smaller of the two Id currents is chosen to be within 10% of the larger.

The current values needed by this method are highly dependent on the voltage limits specified. The voltage limits, however, are nearly universal and will work with nearly any device. The process thus requires Preview to set the voltages, and then uses the voltage data to provide the needed current ranges.

The authors provide derivations for the shift in Vgs that depend only on the gate diode's active resistance and Rs. The form of these expressions cannot be reduced to Rs=xxx, so the measured and computed delta Vgs values are optimized until a suitable Rs is found.

In order for this method to work, the gate diode characteristics Is and N must first be known. These are characterized in the gate_diode measurement. Thus three setups are required for this technique: the Yang_Long_Preview, the gate_diode, and the Yang_Long_Method.

Source_Resistance / Yang_Long_Preview

This setup measures both Ig and Id as a function of Vg, and displays the results on a log scale. The setup is used to ensure that the current input values chosen fall within the limits prescribed in the original paper by Yang and Long.

In this setup the values for several Yang_Long method variables will be determined and set in the model variable table.

The input variables configure the FET as shown in the following figure.
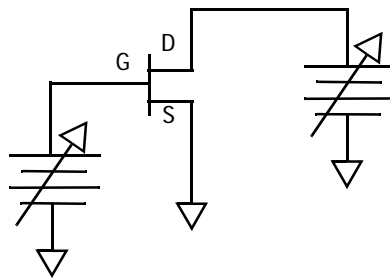
Figure 117   FET Bias Configuration for Yang_Long_Preview Measurement

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements, except set Integ Time to L. Then return to this chapter and continue.

Initializing the Input Variable Values

1 Select **Source_Resistance > Yang_Long_Preview**. and observe the inputs vg and vd (see the following figure).


```
    Input: vg                      Input: vd
        Mode: V                        Mode: V
      + Node: G                      + Node: D
      − Node: GROUND                 − Node: GROUND
        Unit: SMU1                     Unit: SMU2
  Compliance: IGS_MAX           Compliance: IDS_MAX
   Sweep Type: LIN               Sweep Type: LIN
  Sweep Order: 1                Sweep Order: 2
        Start: VG_GML_MIN              Start: VDS_YL_MIN
         Stop: VG_GML_MAX               Stop: VDS_YL_MAX
 # of Points: VGS_FINE         # of Points: 2
    Step Size:                    Step Size:
```
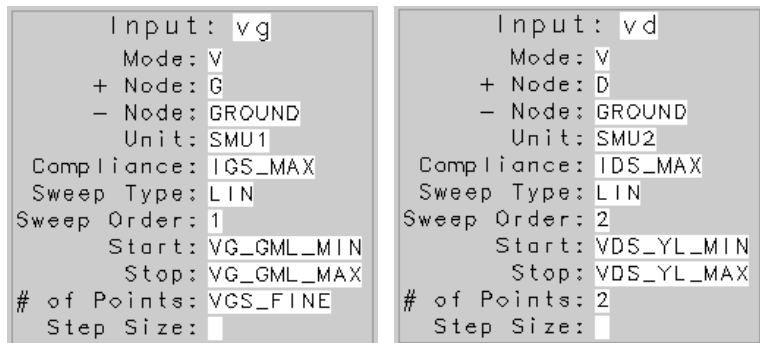
Figure 118   Inputs for Yang_Long_Preview Measurement

- Both the vg and vd Sweep Type are set to LIN for a linear sweep from start to stop values.
- The vg # of Points is set to VGS_FINE to allow clear and accurate definition across the trace.
- The vd # of Points is set to 2 to provide Vd measurements at two closely spaced values.

2 Select **Extract/Optimize** folder, select the **initialize** transform and click **Execute**. The transform selects the appropriate

current values for the device under test and saves them in the variable table.

The VG_GML values should be set to voltages that comply with the methods defined by Yang and Long. Usually the defaults (0.3V for VG_GML_MIN and 0.6V for VG_GML_MAX) are adequate and you will probably not need to change them.

The Rs extraction is done in the linear region of transistor operation. Therefore both VDS_YL_MIN and VDS_YL_MAX are values in the linear region of Vds, at values less than 0.25V. Generally, these Vds default values never need to be changed except in the case of a device with a very sharp knee.

3   Select the **log10ig_id_by_vg** plot and **Display Plot**. The plot should resemble the following figure, with measured traces for both Ig and Id, plotted on a log scale. Sometimes the top end of the measured range may show some resistive effects: this will be windowed out in the extraction procedure later.
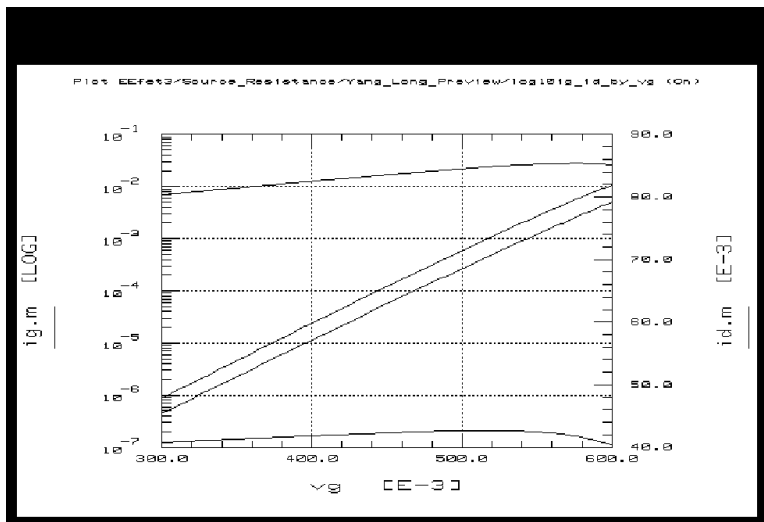
Figure 119   Example Yang_Long_Preview log10ig_id_by_vg Plot

The data on this plot is used to select current values to be used in the Yang_Long_Method setup. You will use the data to select the minimum and maximum values for IGS_YL and IDS_YL, which are used in the extraction of Rs.

4   Using the measured Ids traces, select lower and upper values for Ids that are within ten percent of each other, somewhere in the middle of the two traces. Select round numbers that are within the settability limits of the DC source/monitor. Enter your chosen values in response to the prompts for IDS_YL_MIN and IDS_YL_MAX. (The default value for IDS_YL_MIN is 0.058 amp, and for IDS_YL_MAX is 0.060 amp.)

5   To select the values for Igs, pick values in the region of Igs where the trace is linear. Choose two values that are 50 to 100 times smaller than the Ids values you have chosen. Enter your chosen values in response to the prompts for IGS_YL_MIN and IGS_YL_MAX. (The default value for IGS_YL_MIN is 0.5E−4, and for IGS_YL_MAX is 0.5E−5.)

When you have input all the necessary values, the transform sets numerical values for the relevant variables, and lists them in the model variable table and the UNIX standard output window.

6   To save the plotted data, choose **File > Save As**.

Source_Resistance / gate_diode

This setup measures swept gate current as a function of gate voltage, with Vds constant at zero, and displays the results on a log scale. This will be used for characterization of the gate diode forward conduction parameters Is and N. The dynamic resistance attributable to the gate diode formed by the Schottky contact is important to the extraction of all contact resistances.

The FET configuration is illustrated in the following figure.
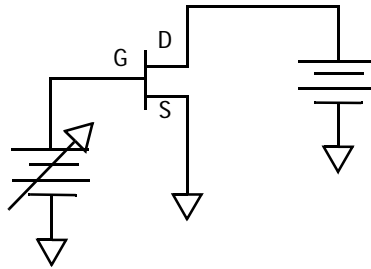
Figure 120   FET Bias Configuration for gate_diode Measurement

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

Setting the Input Variables

1 From the DUT/Setup panel select the gate_diode setup and observe the vg and vd inputs. The following figure illustrates the inputs used in this measurement.
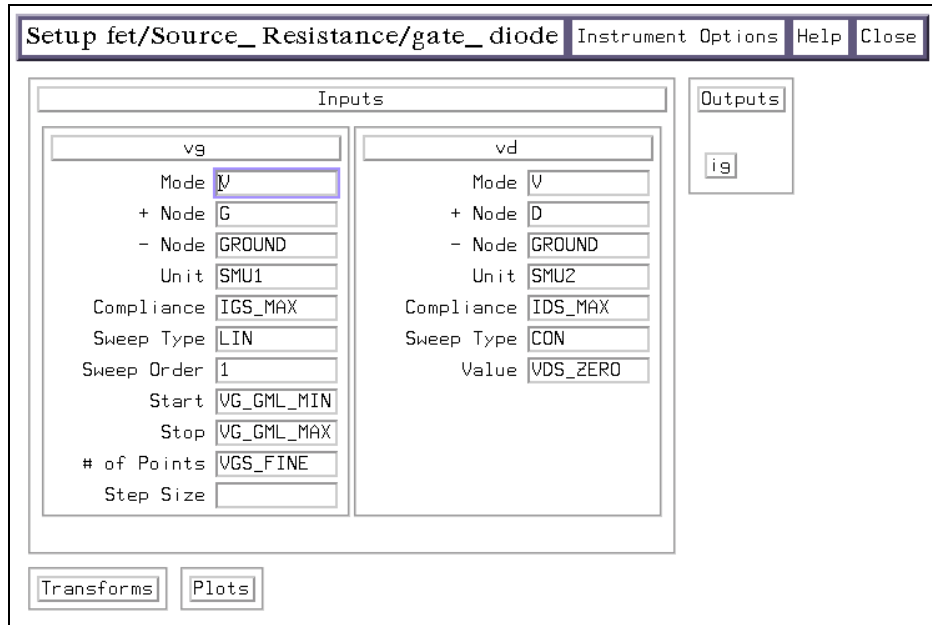
Figure 121 Inputs for gate_diode Measurement

- The vg inputs are the same as in the Yang_Long_Preview measurement, with the gate voltage set to sweep over the voltage range from VG_GML_MIN to VG_GML_MAX.
- vd is set to a CONstant voltage of 0.0.

Measuring and Plotting

1 Select the **log_ig_vg** plot, and **Display Plot**. then click **Measure**. The plot should resemble the following figure, with Ig plotted on a log scale vs. Vg.

2 The transform to compute the value of Ig will be performed in the extraction part of the procedures.
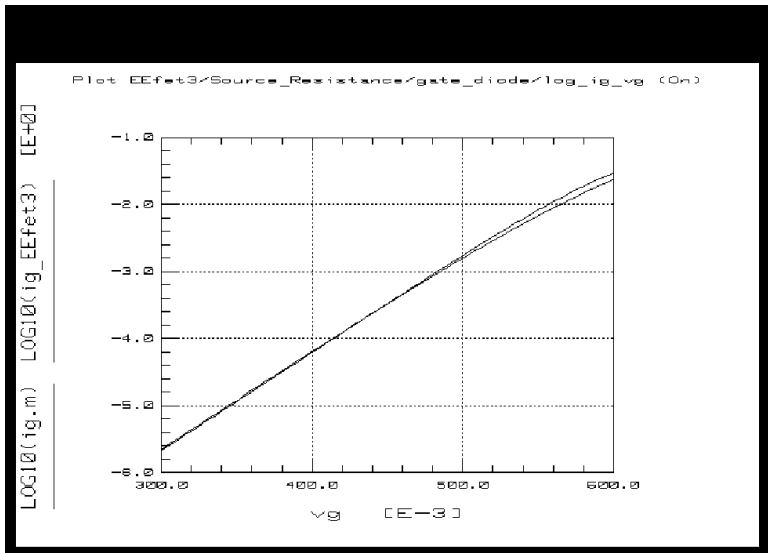
Figure 122   Example gate_diode log_ig_vg Plot

3 Optionally, to save the plotted data, choose **File > Save As**.

4 Close the plot.

Source_Resistance / Yang_Long_Method

This procedure uses Ids and Igs values derived from the results of the Yang_Long_Preview measurement. The setup drives two different relatively small amounts of drain current into the device. It sweeps Igs values that are 50 to 100 times smaller than Ids, to give measured Vgs values that differ slightly: the difference is due to the common lead resistance.

Device Configuration: The input variables configure the FET as shown in the following figure.
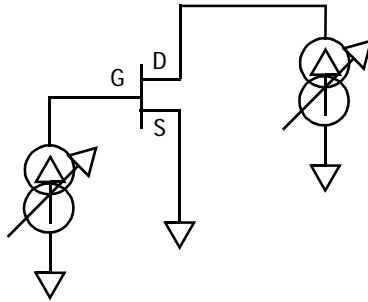
Figure 123 FET Bias Configuration for Yang_Long_Method Measure-ment

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

The Input Variables

1 To view the input variables for this measurement, from the DUT/Setup panel, select the Yang_Long_Method setup and

observe the vg and vd inputs. The following figure illustrates the inputs used in this measurement.
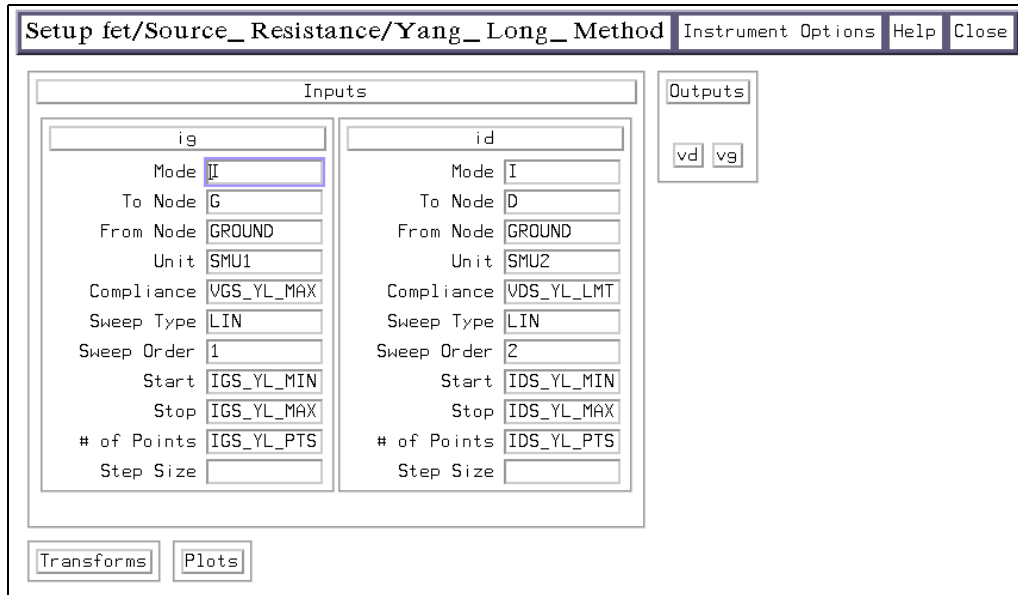

Figure 124   Inputs for Yang_Long_Method Measurement

- Note that for this current-controlled measurement, the Compliances are voltage values.

- The ig Sweep Type is set to LIN for a linear gate current sweep from IGS_YL_MIN to IGS_YL_MAX. These are the extremely small positive voltage values you selected from the measured Yang_Long_Preview data. The IGS_YL_PTS default value of 11 is adequate.

- The id Sweep Type is set to LIN for a linear sweep measured at the IDS_YL_MIN and IDS_YL_MAX values you selected from the measured Yang_Long_Preview data. The IDS_YL_PTS value is always set to 2 for two swept traces.

Measuring and Plotting

1 From the Plots folder (for the Yang_Long_Method setup), select the **delta_m_s plot**, and **Display Plot**, then click **Measure**. The plot should resemble the following figure.
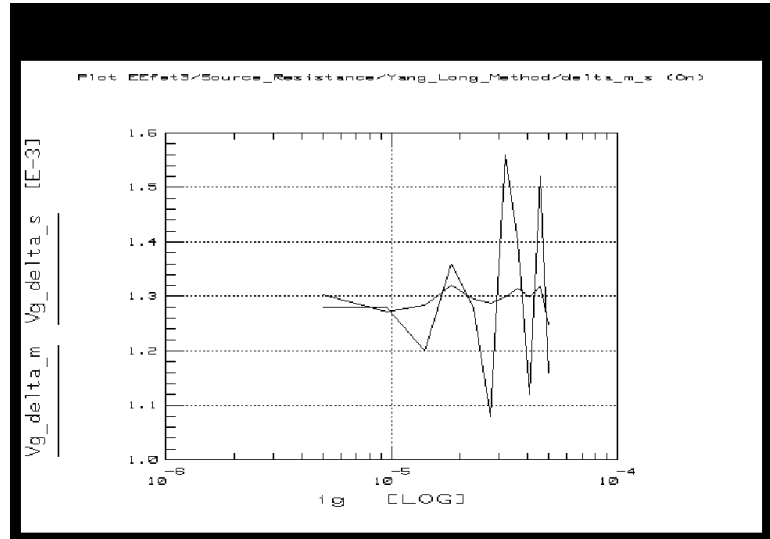
Figure 125   Example Yang_Long_Method delta_m_s Plot

2 Select the **opt_rs** transform and **Execute**.

3 Optionally, to save the plotted data, choose **File > Save As**.

4 Close the plot.

Parasitics Measurements: The cold_fet and package DUTs

The extractions for intrinsic and extrinsic parasitics are performed on S-parameter data measured with a calibrated network analyzer. These measurements are made in the following setups:

- cold_fet/rg_rd
- package/Arnold_Golio

The cold_fet/rg_rd setup

The Rg and Rd contact resistances are extracted from cold FET measurements made with the device strongly forward-biased in both the gate-drain and gate-source regions. The method is similar to the work described by Dambrine, Cappy, et al.

Successful extraction of certain extrinsic parasitic elements is dependent on forcing the active device under test to a bias condition where its linear equivalent circuit reduces to a very simple form. While the Curtice model uses the "cold FET" method (Vds=0.0V, Vgs=0.0V) to optimize the contact resistances and parasitic inductances of the bond wires, further simplification can be achieved by strongly forward biasing the gate-source and gate-drain regions of the FET. This procedure is based on this premise.

Under conditions of strong forward gate bias the intrinsic charge model of the FET is shunted by small resistances of the forward active diode in the gate and the fully open channel between the drain and source. Under these circumstances the reactive behavior of the device is attributable to the lead inductances of the device. Both the drain and gate resistances are separable if the value of Rs is known.

This setup is substantially the same as the work of Dambrine and Cappy, though it does not exactly track their work (see "Contact Parasitic Resistance and Inductance Method" on page 398). The FET is forward-biased at three to five points. The extrinsic elements are constant while the gate and channel characteristics change.

Device Configuration: The following figure illustrates the device configuration for this measurement. For S-parameter measurements, a DC-biased RF signal is applied to the device gate from the network analyzer system source via the bias networks.
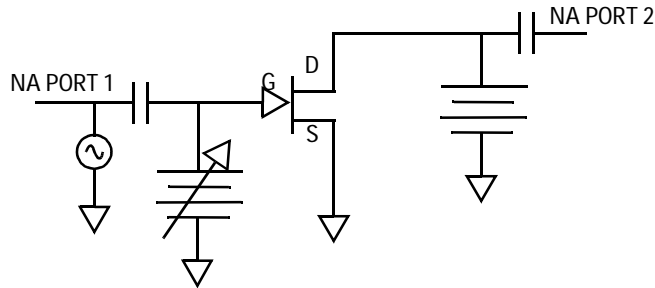
Figure 126 FET Bias Configuration for rg_rd Measurement

Defining the Instrument States

You will need to set the instrument states for both the DC source/monitor and the network analyzer.

For the DC source/monitor, use the same settings as you did for the device preview measurements. Calibrate the network analyzer. In the instrument options for the network analyzer, be sure to set the Cal Set No to the cal set or register where you save your calibration.

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

You will use these instrument states in all the setups that make S-parameter measurements with the network analyzer. Then return to this chapter and continue.

Setting the Input Variables

1 Select the **rg_rd** setup to view the inputs and outputs. Notice The frequency input is set to LIN for a linear sweep over the calibrated network analyzer frequency range.

2 Click the **Model Variables** tab and set *FREQ_START*, FREQ_STOP, and *FREQ_POINTS* to the start frequency, stop frequency, and number of frequency points you used for your network analyzer calibration.

Notice the input vg is set to sweep at three specific values that will heavily forward bias the gate diode.

Notice the input vd is set to CON for a constant value of VDS_ZERO.

Measuring and Plotting

1 In the rg_rd setup select the **im_z12** plot, and **Display Plot**. then click **Measure**. The plot should resemble the following figure. The data plotted is the imaginary part of the intrinsic Z12, derived from S12, which will be used in the extraction

procedures to give preliminary values for the bond inductances, as well as the Rg and Rd values.
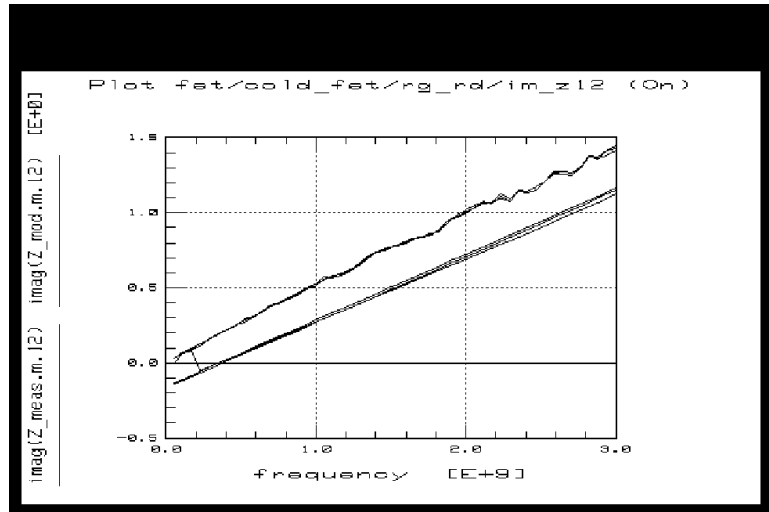
Figure 127   Example im_z12 Plot in the rg_rd Setup

2 Close the plots.

Package Parasitics: the package DUT

RF parameter extraction and large-signal model generation require the reduction of a large number of S-parameter sets into an equivalent circuit format. Although numerical optimization techniques have often been used for extracting these small-signal models, their problems with uniqueness and long execution times make them impractical here. Fast techniques for extracting complete small-signal models have been described previously but all require the additional RF characterization of the cold FET in order to extract parasitic inductance. This offers preliminary inductance values, but the values are not completely accurate since the model has not been derived solely from the data it is asked to model.

The method described here is derived from the work of Arnold and Golio et al (see "Extraction of Package Parasitic and Intrinsic FET Equivalent Circuit Values" on page 398). It requires only RF characterization at the device operating point and knowledge of the parasitic resistances. The technique has been successfully applied to GaAs FETs and HEMTs. After determining the parasitic resistances from DC measurements, the remaining elements are computed directly from a single set of measured S-parameters. The method is very fast, and its results provide an excellent match to measured S-parameters throughout the saturation region.

The package/Arnold_Golio setup

The package resistance and inductance parasitics are extracted from S-parameter data measured across Vgs with Vds at the nominal operating point. It is particularly important to make sure the nominal operating conditions are used for extractions of contact resistance and parasitic parameters, otherwise errors are caused by bias dependencies that are not modeled.

This setup measures S-parameters at values of Vgs from pinchoff to Idss at the nominal Vds operating bias point. This large range of Vgs values allows the largest intrinsic model variations possible. Large variations in the FET element values increase the impact of the static package parasitics, making them easier to extract in the later procedures. The accuracy of this measurement is very dependent on a good network analyzer calibration.

Device configuration:
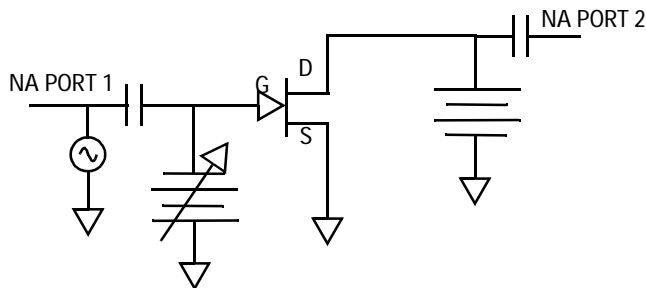
Figure 128   FET Bias Configuration for Arnold_Golio Measurement

Defining the Instrument States

Use the same instrument state settings for this procedure as you did for the rg_rd measurement, for both the DC source/monitor and the network analyzer. *The instrument states need to be set independently for each setup.*

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Then return to this chapter and continue.

Setting the Input Variables

1 Select the Arnold_Golio setup and observe the inputs. The following figure illustrates the inputs used in this measurement.
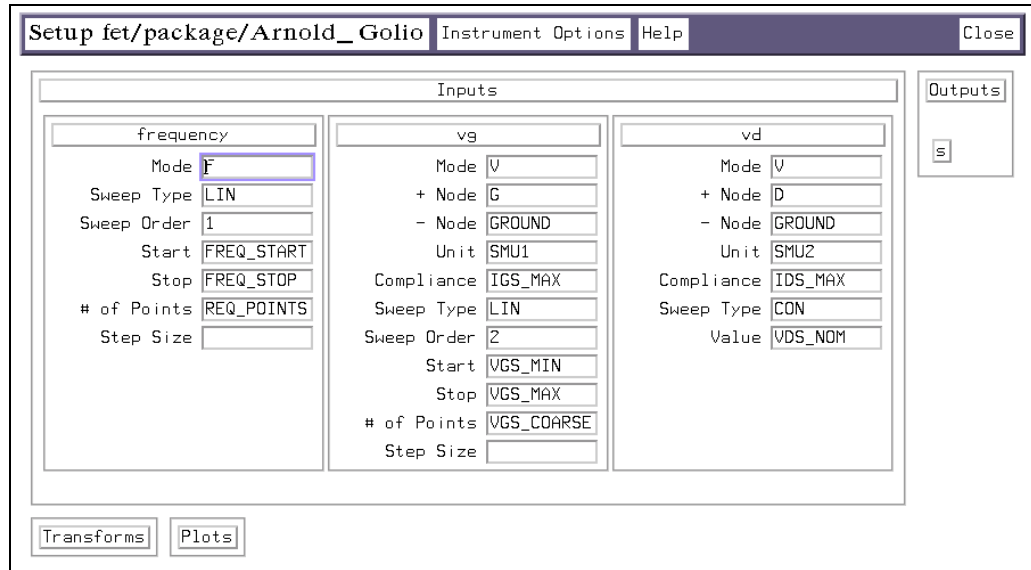
Figure 129   Inputs for Arnold_Golio Measurement

- The frequency inputs are set the same as for the rg_rd measurement.
- The vg value of VGS_MIN (which you set in the device preview measurements) is a value that is more negative than the device pinchoff voltage at the nominal operating voltage of Vds.
- VGS_MAX (set in device preview) is the highest (most positive) Vgs value where the device can be operated quiescently at VDS_MAX.
- VGS_COARSE is a number of points between 5 and 15.

- Vd is set for a CONstant value of VDS_NOM, the nominal operating voltage of the device. VDS_NOM has not been used in any of the measurements so far, so you may need to set the value in the variable table.

Measuring and Plotting

1 From the Arnold_Golio setup, open the **Plots** folder and **Display Plots**, then click **Measure**.

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plots.

The dc_iv DUT

The setups in the dc_iv DUT are used to obtain data for extracting DC-IV parameters. The setups used for these measurements are:

- ig_Is_N
- id_vgs_at_vdso
- id_vgs
- id_vds

The measurement procedures are straightforward, and in most cases are duplicates of measurements you have made in device preview or as part of the source resistance measurements. Therefore, in the majority of cases, you have already defined the measurement input values in the model variable table or in response to prompts, and will not need to modify the inputs. However, the inputs are illustrated and explained for clarification of how each measurement is made. You will need to define the instrument states in each setup.

To prevent possible bias oscillation, put the network analyzer in hold sweep mode.

dc_iv / ig_Is_N (Measuring)

This is the same measurement made in the gate_diode setup as part of the source resistance measurements. It measures swept gate current as a function of gate voltage with Vds constant at zero, and displays the results on a log scale.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

The Input Variables

1 From the DUT/Setup panel, select the **ig_Is_N** setup and observe the inputs, as illustrated in the following figure.
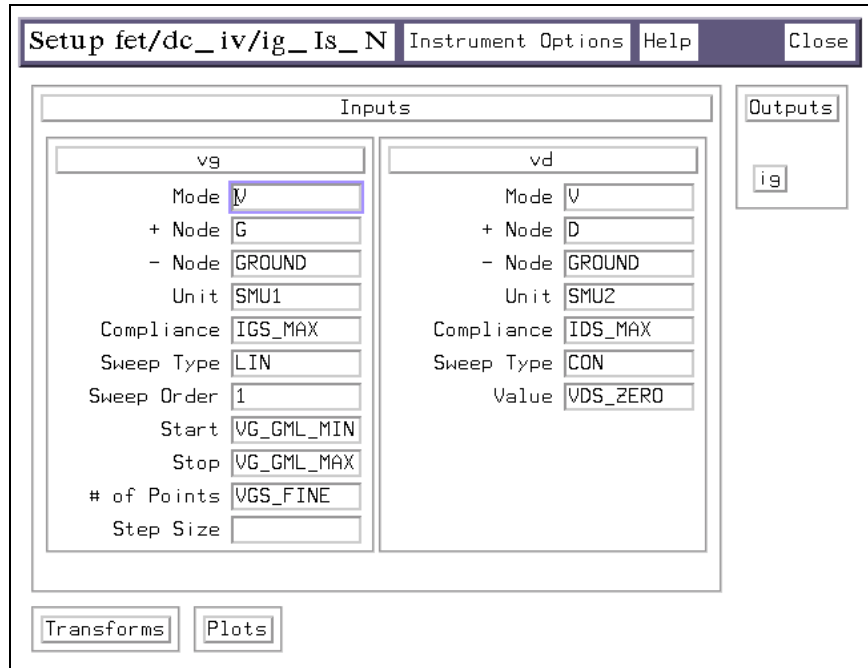
Figure 130 Inputs for ig_Is_N Measurement

- The gate voltage is set to sweep over the same Gummel range as in the gate_diode measurement.
- The drain voltage is set to VDS_ZERO.

Measuring and Plotting

1 Select the **log_ig_vg** plot, and **Display Plot** and then click
Measure. The plot should resemble the following figure, with
Ig plotted on a log scale vs. Vg.

Figure 131   Example log_ig_vg Plot in the ig_Is_N Setup

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plot.

dc_iv / id_vgs_at_vdso (Measuring)

This is the same as the ids_vgs_at_vdso measurement you made
as part of the device preview measurements. This measures all
the values of Vg with Vds at a constant value of Vdso. It covers
all regions of device operation from subthreshold up to the high
current region.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown
next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

The Input Variables

1 From the dc_iv DUT, select the **id_vgs_at_vdso** setup and observe the inputs as illustrated in the following figure.
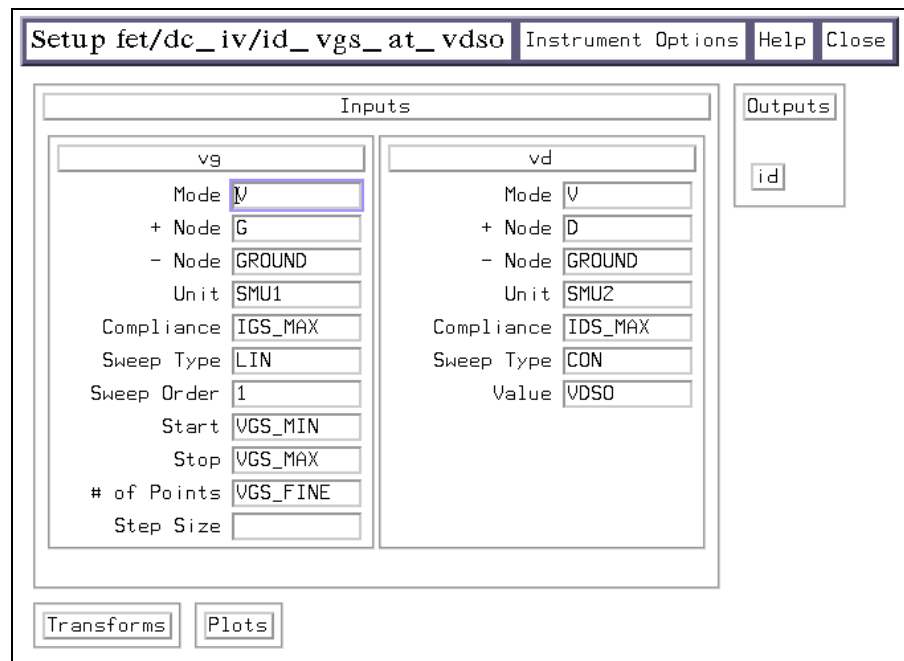
Figure 132   Inputs for id_vgs_at_vdso Measurement

- The gate voltage is set to sweep across the device operating range from minimum to maximum.
- The drain voltage is set to a constant value equal to Vdso, the point where the model equations simplify.

Measuring and Plotting

1 Select the id_vg plot, and **Display Plot**. then click **Measure**. The plot should resemble the following figure with a single curve IV plot.
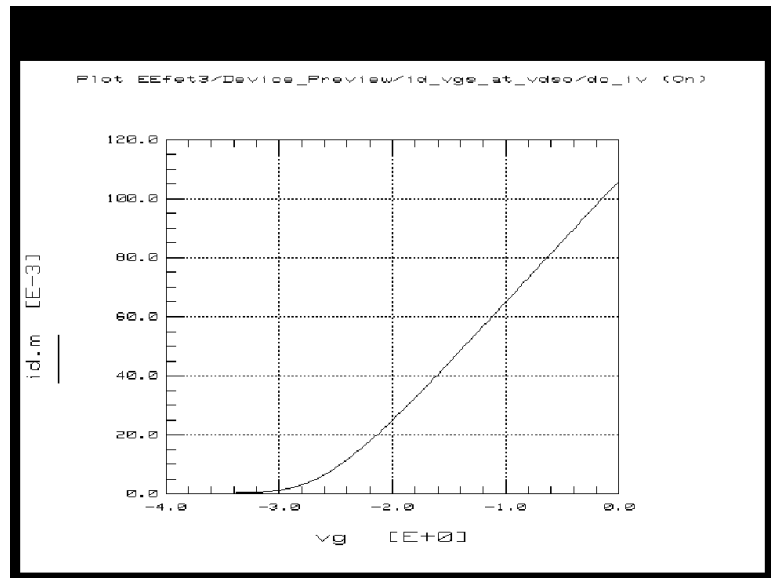
Figure 133   Example id_vg Plot in the id_vgs_at_vdso Setup

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plot.

dc_iv / id_vgs (Measuring)

This procedure measures drain current with respect to drain voltage at several values of gate voltage. The drain voltage is swept from 0V to the upper limit of its operating range.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

The Input Variables

Select the **id_vgs** setup and observe the inputs. The following figure illustrates the inputs for this measurement.
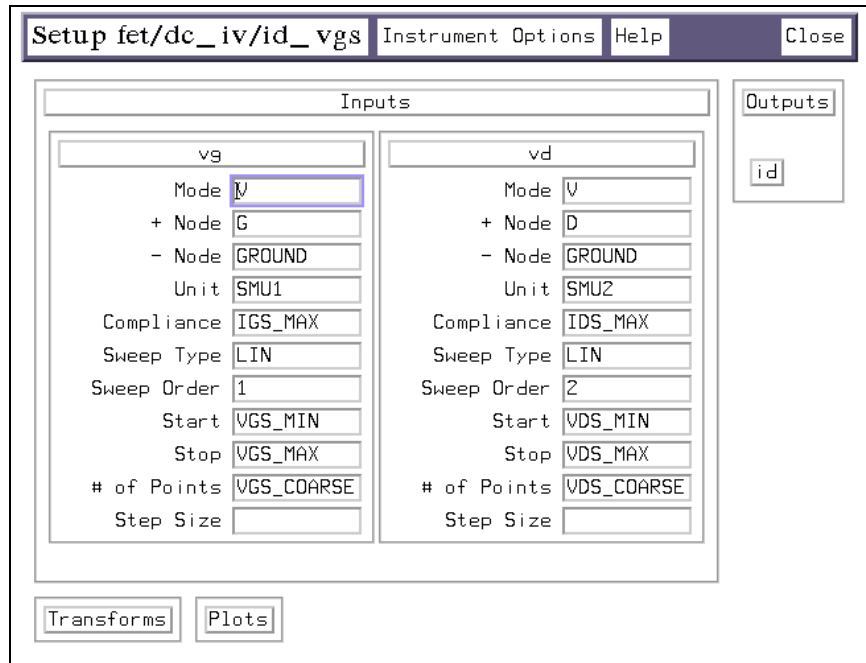
Figure 134 Inputs for id_vgs Measurement

- The gate voltage is set to sweep across the device operating range from minimum to maximum.
- The drain voltage is set to sweep from its minimum value to its maximum safe value.

Measuring and Plotting

1 Select the **id_vg** plot, and **Display Plot** then click **Measure**. The plot should resemble the following figure.
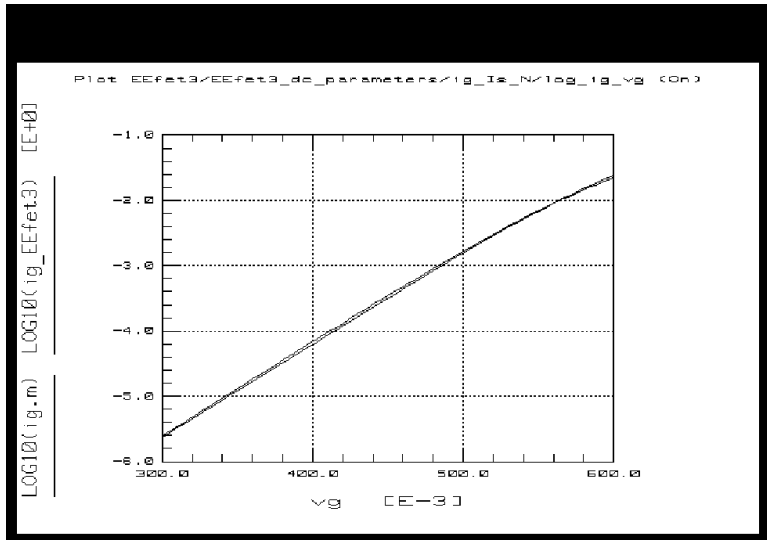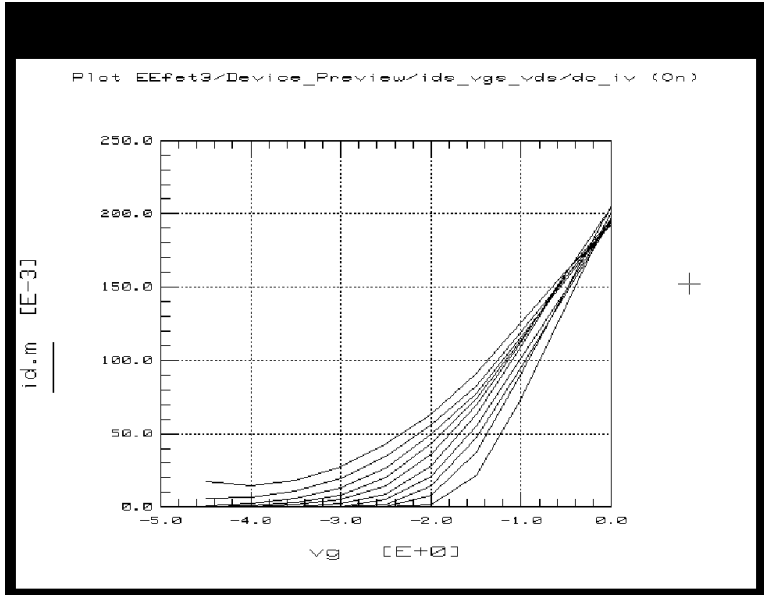

Figure 135 Example id_vg Plot in the id_vgs Setup

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plots.

dc_iv / id_vds (Measuring)

This is the same as the ids_vds_vgs measurement you made as part of the device preview measurements. The drain voltage is swept from 0V to the upper limit of its operating range. The measurement covers the overall region of device operation, and plots the characteristic IV curves vs. Vds. It uses the same inputs as the id_vgs measurement, except with the sweep order for vg and vd reversed.

Defining the DC Source/Monitor Instrument State

Follow the steps described in the appropriate chapter shown next:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same settings as you did for the device preview measurements. Then return to this chapter and continue.

The Input Variables

1 Select the **id_vds** setup and observe the inputs, as illustrated in the following figure.
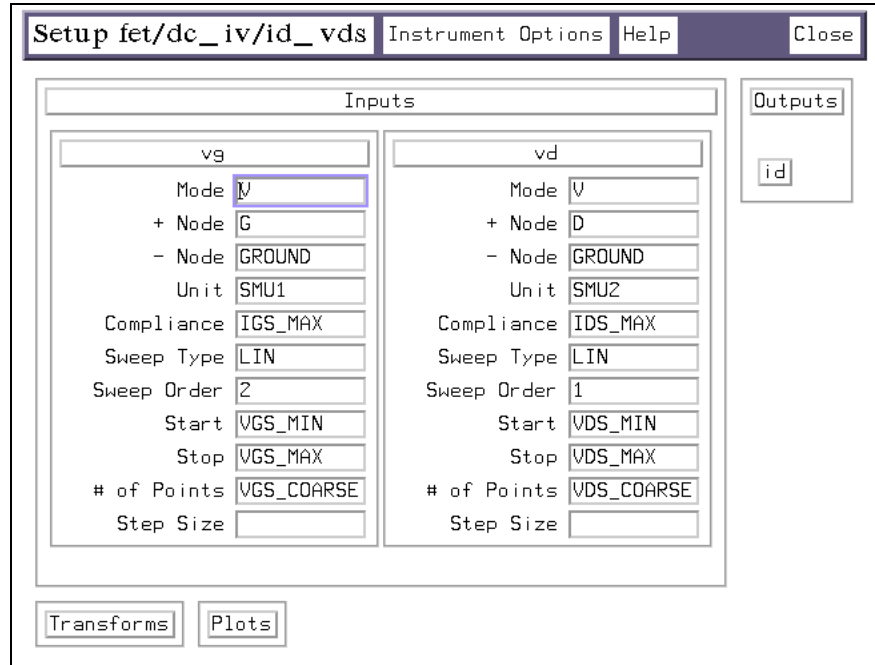
Figure 136   Inputs for the id_vds Measurement

- The gate and drain voltage values are set the same as for the id_vgs measurement.
- Note that the Sweep Order for vg is set to 2, and for vd is set to 1 to make the drain voltage the primary sweep.

Measuring and Plotting

1 From the Plots folder select the **id_vd** plot, and **Display Plot** then click **Measure**. The plot should resemble the following figure, displaying the device characteristic curves.
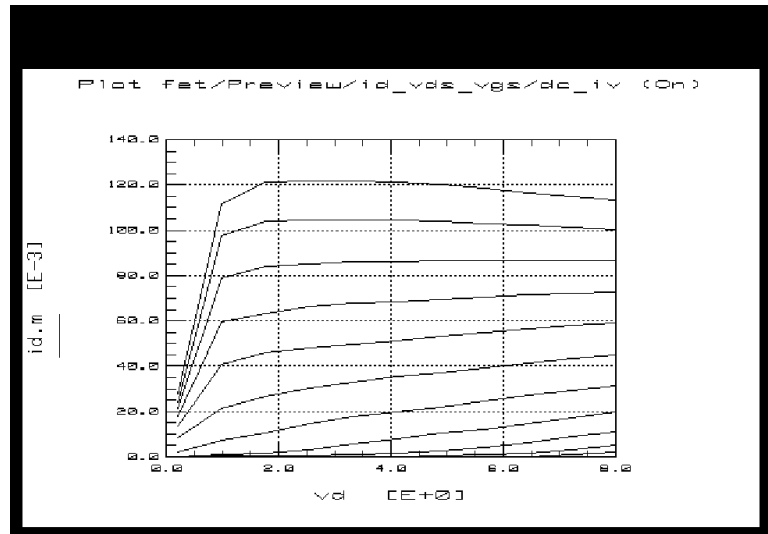
Figure 137   Example id_vd Plot in the id_vds Setup

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plot.

The AC_at_VDSO DUT

This DUT is used for making AC charge and dispersion parameter measurements. Two different S-parameter measurements are necessary for the extractions of the AC charge and dispersion parameters. The first measurement is performed with Vds set to a constant bias value of Vdso. The second has Vds actively biased across the device operating range. It is essential that the measurements be performed in the order listed, and that the extractions listed in the same DUT groupings with the measurement setups be performed from

those corresponding setups. The extraction procedures later in the chapter will make clear which measurement setup is to be used for which extraction.

Make sure the network analyzer is in continuous sweep mode.

AC_at_VDSO / Meas_Spars

This setup measures S-parameters at values of Vgs from pinchoff to Vgs_max with Vds at Vdso, the point where the model equations simplify. The large range of Vgs values allows the largest intrinsic model variations possible, increasing the impact of the charge parameters and making them easier to extract. A good network analyzer calibration is important to the accuracy of this measurement and the subsequent charge parameter extractions.

Device Configuration: The following figure illustrates the device configuration for this measurement. A DC-biased RF signal is applied to the device gate from the network analyzer system source via the bias networks.
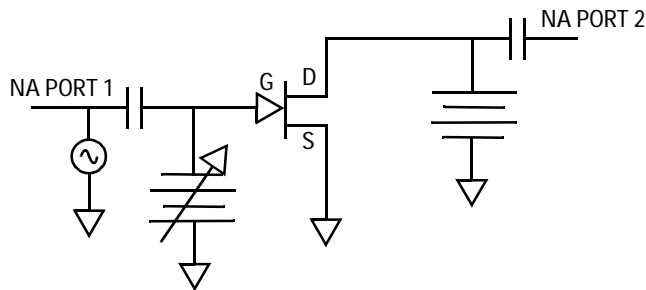
Figure 138   FET Bias Configuration for the First Meas_Spars Measurement

Defining the Instrument States

You need to set the instrument states for both the DC source/monitor and the network analyzer.

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same instrument state settings for this procedure as you did for the rg_rd and Arnold_Golio measurements. Then return to this chapter and continue.

The Input Variables

From the AC_at_VDSO DUT, select the **Meas_Spars** setup and observe the inputs. The following figure illustrates the inputs used in this measurement.
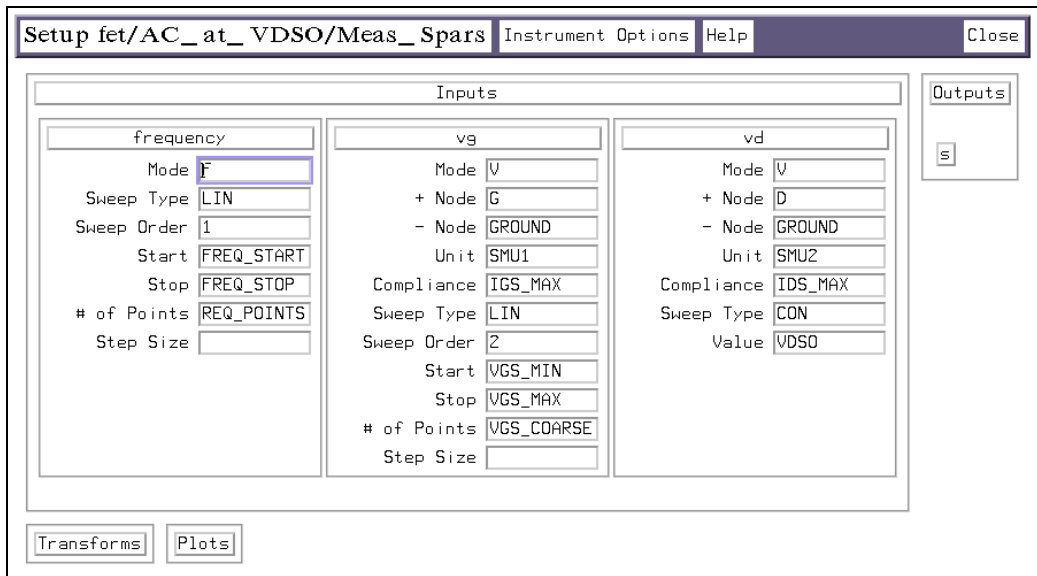
Figure 139   Inputs for the First Meas_Spars Measurement

- The frequency inputs are set the same for all S-parameter measurements in this model, corresponding with the frequency settings of the network analyzer calibration.

- The gate voltage is set to sweep across the device operating range from minimum to maximum.

- The vg # of Points is set to VGS_COARSE. This ensures that the data taken is the least that will yield good measurement results with S-parameter plots that are understandable. Using too many measurement points causes the extractions to be very time-consuming and the plots to be confusing.

- The drain voltage is set to a CONstant value equal to Vdso, the point where the model equations simplify.

Measuring and Plotting

1 With the Meas_Spars setup selected, **Display Plots** then click **Measure**.

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plots.

About the Other Setups in the AC_at_VDSO DUT

Note that no measurements are performed in the other setups in this DUT. These setups will be used later, in the extraction procedures, based on the data measured in this first Meas_Spars setup. However, it is important that the DC input variables for vg and vd be set exactly the same as in the first Meas_Spars setup, with one exception. The exception is that the vg Sweep Order is set to 1 instead of 2, because there is no frequency sweep.

AC_all / Meas_Spars

This setup measures S-parameters at values of Vgs from pinchoff to Vgs_max and values of Vds swept across the device operating range. Again, a good network analyzer calibration is essential: if the calibration is incorrect or has drifted, the modeled data will be impossible to fit to the measured data.

Device Configuration:
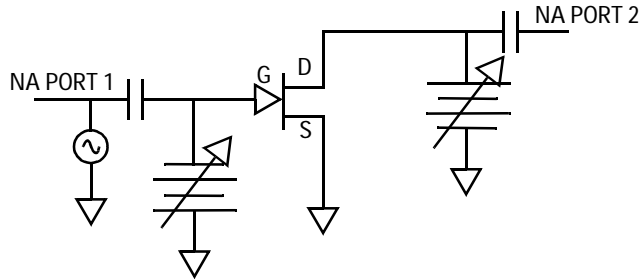
Figure 140   FET Bias Configuration for the Second Meas_Spars Measurement

Defining the Instrument States

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Defining the DC Source/Monitor Instrument State" on page 621 (for an Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Use the same DC source/monitor and network analyzer settings as you did for the first Meas_Spars and the other S-parameter measurements. Then return to this chapter and continue.

The Input Variables

From the AC_all DUT, select the **Meas_Spars** setup and observe the inputs. The following figure illustrates the inputs used in this measurement.
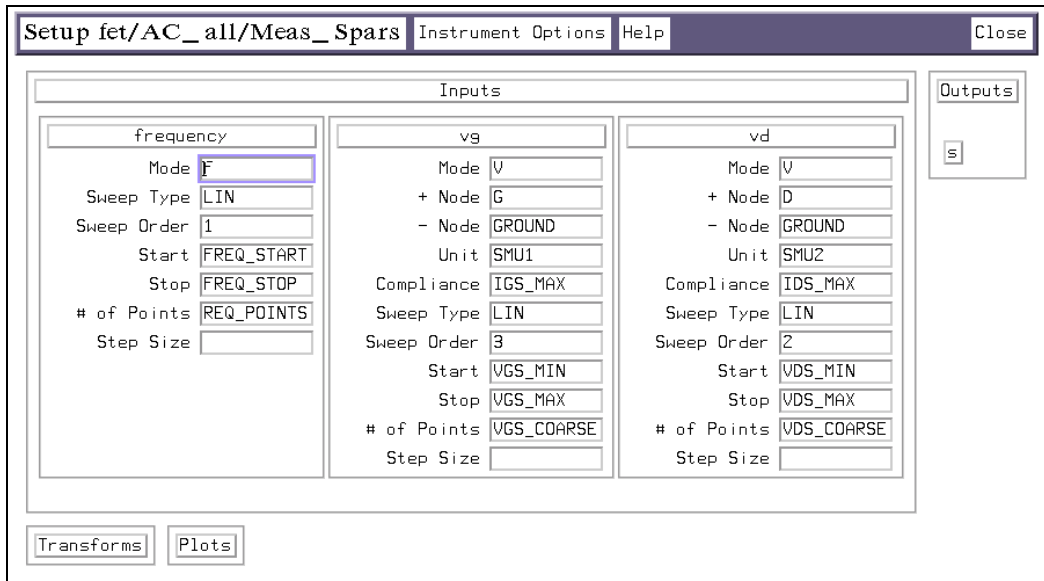
Figure 141 Inputs for the Second Meas_Spars Measurement

- The frequency inputs are set the same as for the other S-parameter measurements.
- The gate voltage is set to sweep across the device operating range from minimum to maximum.
- The drain voltage is set to sweep from its minimum value to its maximum safe value.
- The # of POINTS for both vg and vd is set to COARSE to allow good measurement results without excessive data-taking.

Measuring and Plotting

1 With the Meas_Spars setup selected, **Display Plots** then click **Measure**.

2 Optionally, to save the plotted data, choose **File > Save As**.

3 Close the plots.

About the Other Setups in the AC_all DUT

Note that no measurements are performed in the other setups in this DUT. They will be used later, in the extraction procedures, based on the data measured in the second Meas_Spars setup. However, it is important that the DC input variables for vg and vd be set exactly the same as in the second Meas_Spars setup, with one exception. The exception is the Sweep Order, which is set to 1 for vg and 2 for vd, because there is no primary frequency sweep.

Storing the Measured Data

This procedure stores the data you measured for this particular device. You can then use the data to perform the extractions, to compare the simulated data with the measured data, and as a measurement record for comparison with other devices.

If you will be measuring large numbers of devices, it is worthwhile to organize your directory structure according to device type, geometry, and batch or wafer.

1 Decide where in your directory structure you intend to store this data file.

2 From the IC-CAP/Main window select **Model**, then select **File > Save As** from the pulldown menu. A dialog box is displayed.

3 Choose a filename that will clearly identify the measured device and the type of information stored, such as fet2_meas_12aug95. Type the directory path and the filename into the dialog box.

4 Click **OK** or press Enter to store the data and close the dialog box.

5 When you want to use this data for an extraction at some future date, remember to change directories in IC-CAP by selecting **Windows > Main Window** and **File > Change Directory**... (See "Accessing Data from a Previous Measurement" on page 569.)

This concludes the measurement procedures. The extraction procedures follow.

Parameter Extractions

Once the measurements have been performed and a complete set of data collected for a particular device, the extraction can proceed. If you saved the measured data, you can perform an extraction at any later time. The Xtract macro can be used for an unattended extraction, or each extraction step can be processed individually to allow complete process control. The procedures in the following pages explain how to extract the parameters individually.

Displaying the Parameters Table

If the Parameters table is not already displayed on the screen, select **Model Parameters** from the Model window, and select **Detach**.... Move the Parameters table to a convenient corner of the screen: you will be using it throughout the extractions.

Extracting the Source Parasitic Resistance Rs

The Yang and Long method computes the value of Rs from the change in Vgs due to the application of two drain currents, which are 50 to 100 times greater than Ig. The two drain currents are within the drain's linear IV region (Vds < 0.25 volts). The smaller of the two Id currents is within 10% of the larger. The values were set in the source resistance measurement procedures.

The authors Yang and Long provide derivations for the shift in Vgs that depend only on the gate diode's active resistance and Rs. The form of these expressions cannot be reduced to Rs=xxx, so the measured and computed delta Vgs values are optimized until a suitable Rs is found.

In order for this method to work, the gate diode characteristics Is and N must first be known. These are extracted from the data measured in the gate_diode setup. Thus two setups are required for this extraction: the gate_diode and the Yang_Long_Method setups.

Extracting Is and N

1 In the Parameters table, locate the values for EEFET3.rs, Is, and n.

2 From the DUT/Setup panel, select the **gate_diode** setup, and select the **Plots** folder.

3 From the Plots folder select **log_ig_vg**, and **Display Plot**. The plot shows Ig plotted on a log scale vs Vg.

4 From the Extract/Optimize folder, select the **opt_is_n** transform.

5 Click the **Parameters** tab to view the lower and upper optimization limits for EEFET3.Is and .n.

6 Click the Inputs tab to view the upper and lower bounds for Vgs that should limit the optimization range to the linear portion of the IV curve. If for some reason these limits are not sufficient, use the next step to establish more restrictive limits.

7 Click the **Options** tab to view the X_LOW, X_HIGH, Y_LOW, and Y_HIGH functions. These are optimization limits you can set from the data plot to select a portion of the trace for the optimization. The gate diode parameters should be optimized over only the linear portion of the trace, so use the X-low/X-high function as follows:

   • Select setup **gate_diode**, then select **Setup Variables** folder. A variable table will appear, with current values for X-LOW and X-HIGH.

   • From the plotted data choose a linear portion of the trace, along the constant slope.
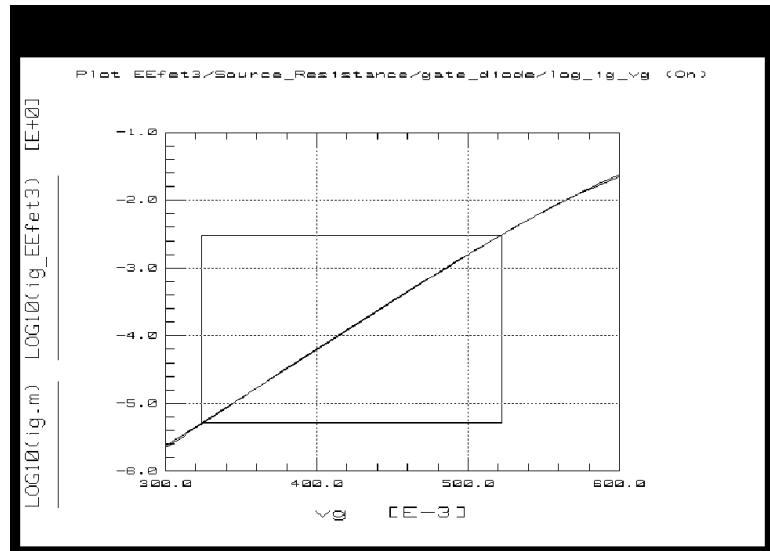
Figure 142   Log Plot of Ig vs. Vg, with X-Low/X-High Values Selected

- • Put the mouse pointer at the low end of the chosen portion and click left, then put the pointer at the high end and click left, to make a box on the trace, as shown in the previous figure.

- • From the plot select **Options > Copy to Variables**. This changes the X_LOW and X_HIGH values to the lower and upper limits you selected on the trace.

8 Click **Execute**. The transform optimizes the plotted data, using the linear portion of the trace you defined with X-low/X-high, and enters the optimized values for Is and n in the Parameters table.

9 Close the plot.

Optimizing Rs

Driving two different relatively small amounts of drain current into a FET and then making swept Vgs measurements vs. Igs gives two sets of measured Vgs values that differ slightly. The

difference is due to the common lead resistance. Measured and simulated delta_Vgs values are optimized in this procedure to yield a reasonable value of the common lead resistance Rs.

NOTE   This extraction generally uses current values within the limits prescribed in Yang and Long's original paper. (These are values you set in the Yang_Long_Preview measurement procedure.) If you chose values outside these limits, warning messages in the extraction phase will alert you to that fact. However, the warnings will not prevent the computations from completion, as you may have a valid reason not to use the specified limits: in this case, ignore the warning messages and use the computed values.

1   Select the **Yang_Long_Method** setup, and **Display Plots**.
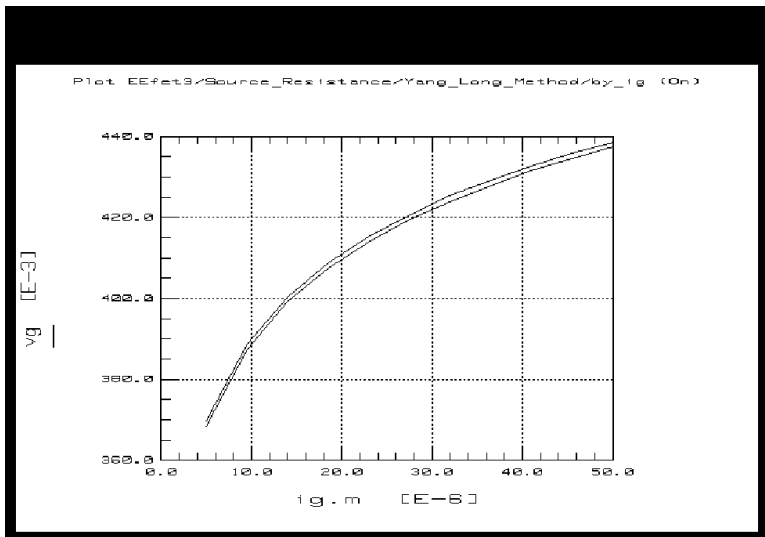
Figure 143   Example Yang_Long_Method by_vgs Plot

2   From the Extract/Optimize folder, select **opt_rs**, and **Execute**. Observe the plots as the traces converge. The by_vgs plot shows the difference between the measured and simulated data.

The opt_rs transform also calls the Vg_delta_m and Vg_delta_s transforms, so you need not perform them separately. The transforms optimize the measured and simulated Vgs shifts for the most accurate source resistance extraction.
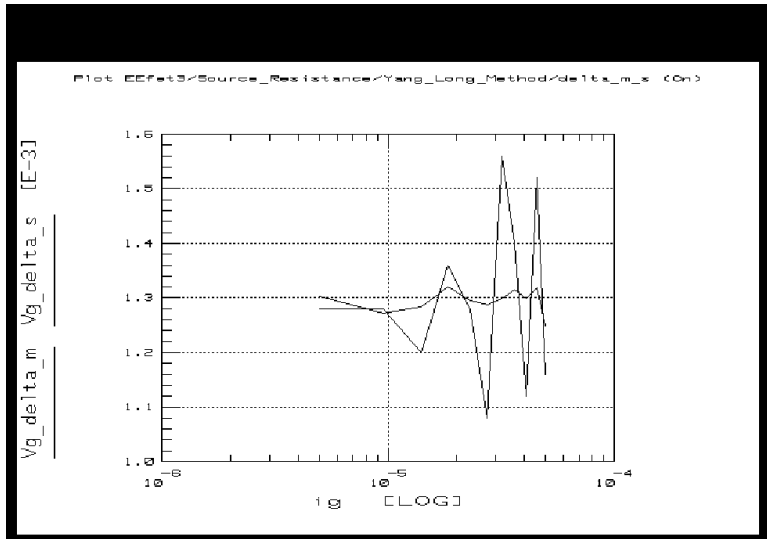
Figure 144   The Computed Shift in Measured and Simulated Vgs Values

The difference in measured and simulated Vgs values is used for the calculation of Rs. The previous figure illustrates the results of the optimization. The difference in Vgs values is caused by the common (source) lead resistance. Although the traces appear very different, note that the scale is very small and the values result in differences of fractions of an ohm.

3 Close the plots.

Extracting the Contact Resistances Rd and Rg

This extraction uses a device setup with the gate-source and gate-drain junctions of the FET strongly forward biased. Under conditions of strong forward gate bias the intrinsic charge model of the FET is shunted by small resistances of the forward active diode in the gate and the fully open channel between the

drain and source. Under these circumstances the reactive behavior of the device is attributable to the lead inductances. Both the drain and gate resistances are separable if the value of Rs is known.

The FET is forward-biased at three to five points. The extrinsic elements are constant while the gate and channel characteristics change. In this extraction the extrinsic elements are optimized while the intrinsic device resistances are allowed to shift. The values of the extrinsic inductance elements are only initial values that will be refined in a later procedure.

This extraction is based on the S-parameter data you measured in the rg_rd setup. A transform in this extraction computes and saves a table of linear equivalent circuit parameters based on the measured S-parameters. It calculates the contact resistance parameters directly from the measured data, and allows you to compare the linear model extracted to the measured S-parameters.

1 In the Parameters table, locate the values for EEFET3.rg and rd.

2 From the DUT/Setup panel, select the **rg_rd** setup.

3 From the Plots folder, select the **im_Z12** plot, and **Display Plot**. The data plotted is the imaginary part of the intrinsic Z12, derived from S12.

4 From the Extract/Optimize folder, select the **auto_extr** transform and **Execute**. This computes the values for the parasitic resistances Rg and Rd. It also calculates initial values for the parasitic inductances Lg, Ld, and Ls, which will be refined in the next procedure. The pad capacitances are defaulted to 0.0 for this procedure, and will be extracted later.

The calculated values for EEFET3.rg and rd will be entered in the Parameters table. Values for Rs, Rd, and Rg, as well as Ls, Lg, and Ld, will be displayed in the UNIX standard output window. An example of the calculated Z-parameter data is illustrated in the following figure.
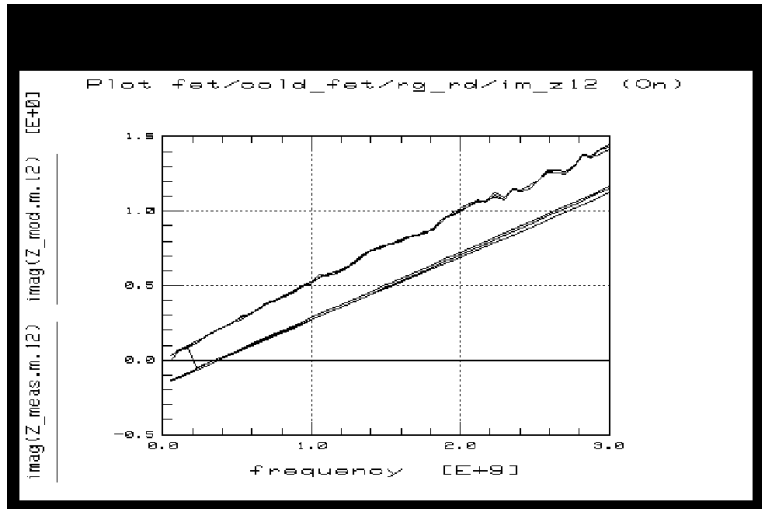
Figure 145   Example im_Z12 Data in the rg_rd Setup

5 Close the plot.

Extracting the Package Parasitics

This extraction is based on the S-parameter data you measured in the Arnold_Golio setup (S-parameters at values of Vgs from pinchoff to Idss at the nominal Vds operating bias point). As the device goes through this wide range of measurement variations the package parasitic inductances remain constant, allowing them to be readily extracted.

A transform in this extraction computes and saves a set of linear equivalent circuit parameters based on the measured S-parameters. A set of modeled S-parameters are simulated based on the linear equivalent circuit. Any differences between the simulated and measured S-parameters are due to the package parasitics or to defects in the network analyzer calibration. (This is why a good network analyzer calibration is

essential for success in this extraction.) A second transform optimizes the measured and simulated S-parameter data to extract the package parasitic inductances.

1 In the Parameters table, locate the values for lg.l, ld.l and ls.l.

2 From the DUT/Setup panel, select the **Arnold_Golio** setup, and **Display Plots**.

3 From the Extract/Optimize folder, select **mdl**, and **Execute**. This computes a set of linear equivalent circuit parameters based on the measured S-parameters, and uses them to simulate a set of modeled S-parameters.

4 Select the transform **opt_pkg**, and **Execute**. This calculates the values for the package parasitic inductances.

   You will see the traces converge on the plots, and the new calculated values for lg.l, ld.l, and ls.l will be entered in the Parameters table. Note that these are final values for the initial inductances calculated in the last procedure.

5 Optionally, you can change the optimization limits. Select **Arnold_Golio** setup, and **Setup Variables**. Modify values in the table as required.

6 Close the plots.

Extracting DC-IV Parameters

The DC-IV extractions start with preliminary estimates of some parameters and begin to establish firm values for some others. Four measurement setups and their measured data are used. The fourth and last DC extraction finalizes the initial values determined in the three previous setups.

Note that the thermal factor Peff has a major impact on the performance of nearly all model parameters. Therefore the initial value for Peff is set so high that it has no impact on model performance.

dc_iv / ig_Is_N (Extracting)

This procedure uses measured gate current data to extract final values for the gate diode characteristics Is and N. (Initial values were found in the source resistance extraction using the gate_diode setup.)

1 Locate the value for EEFET3.peff in the Parameters table. Make sure the Value is set to a high number such as 100, high enough that it will have no effect on model performance.

2 Locate the values for EEFET3.Is and n.

3 From the DUT/Setup panel, select the **ig_Is_N** setup.

4 From the Plots folder, select **log_ig_vg** plot, and **Display Plot**. The graph shows Ig plotted on a log scale vs. Vg.

   The parameters should be optimized over only the linear portion of the trace. The input table contains upper and lower bounds for Vgs that should limit the optimization range to the linear portion of the IV curves. If for some reason these limits are not sufficient, use the X-low/X-high function to define the limits for a window of the linear trace.

5 From the Extract/Optimize folder select **opt_is_n**, and **Execute**. The transform optimizes the plotted data, using the linear portion of the trace you defined with X-low/X-high, and enters the optimized values for Is and n in the Parameters table.

6 Close the plot.

dc_iv / id_vgs_at_vdso (Extracting)

This procedure uses measured Ids data with Vg swept across the device operating range and Vd set to a constant value of Vdso. Thus it covers all regions of device operation from subthreshold up to the high current region, and makes use of the fact that the model equations simplify at Vdso.

This data is used to set the value of Vto and to plot a curve for Gm. From these values the procedure extracts initial values for Vgo and Vdelt, as well as several other FET and HEMT parameters.

1 Leave EEFET3.peff set to a high value.

2 Locate the values for EEFET3.vt and EEFET3.vdelt.

3 From the dc-iv DUT, select the **id_vgs_at_vdso** setup, and **Display Plots**.

4 From the Extract/Optimize folder, select the **initialize** transform and **Execute**. This sets up the software to extract the device-specific parameters.

5 Select **opt_vto_fet3** for a FET, or **opt_vto_hemt1** for a HEMT, and **Execute**. The transform optimizes the plotted data, and enters the optimized values into the Parameters table.

dc_iv / id_vgs (Extracting)

This procedure is used to calculate an initial value for gamma, the parameter that controls the amount Vto shifts with increasing Vds. The final value for gamma will be refined in the last DC extraction.

The control range set for the data optimization is very important in this procedure.

1 Leave EEFET3.peff set to a high value.

2 Locate the value for EEFET3.gamma.

3 From the DUT/Setup panel select the **id_vgs** setup.

4 From the Plots folder, select **id_vg** plot, and **Display Plot**.

5 In the Model window select **Setup Variables**. Note the default values for Y_LOW and Y_HIGH. This control range for the data optimized is important: low values of Ids must be used. Ids values have to be windowed so that only the first 5% to 10% of the Ids data is used, usually between 0.0 and a few milliamps for each trace. The X values should be left at 0 to ensure that all the X-axis data is used in the optimization.

6 From Extract/Optimize folder, select **opt_gamma** and **Execute**. You will see the low end of the traces converge on the plot, and the new calculated value for gamma will be entered in

the Parameters table. The following figure shows an example of the initial optimized gamma data.
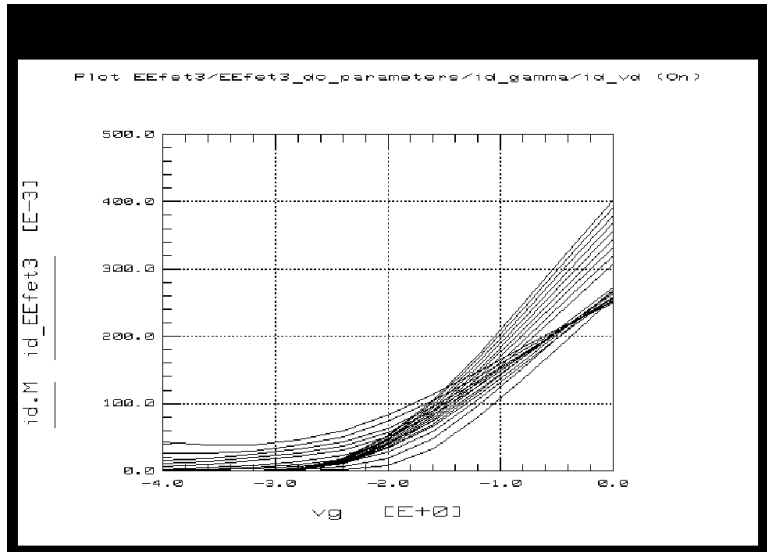
Figure 146 Example Initial Optimized Gamma Data in the id_vgs Setup

Since this is an initial extraction step, the model fit is not ideal. Certain effects will be better modeled in subsequent stages.

7 Close the plot.

dc_ic / id_vds (Extracting)

This procedure extracts final values for parameters initially extracted in the DC-IV extraction procedures. In addition to Vdelt and Gamma, this procedure extracts the parameters Kappa, Vsat, and Peff.

1 Locate the values for EEFET3.vdelt, vsat, and vgo in the Parameters table.

2 From the DUT/Setup panel, select the **id_vds** setup and display the id_vd plot.

3 From the Extract/Optimize folder, select **opt_iv_fet3** or **opt_iv_hemt1**, and **Execute**. The transform optimizes the plotted data, and enters the optimized parameter values in the Parameters table.

Use the X-low/X-high function if desired, to select only a portion of the data for the optimization. This is appropriate if any of the plotted data looks anomalous.

4 Locate the optimized values for EEFET3.gamma, kappa, and peff in the Parameters table. You will notice that the value for EEFET3.peff has changed significantly. The following figure shows an example of the final optimized DC-IV curves.
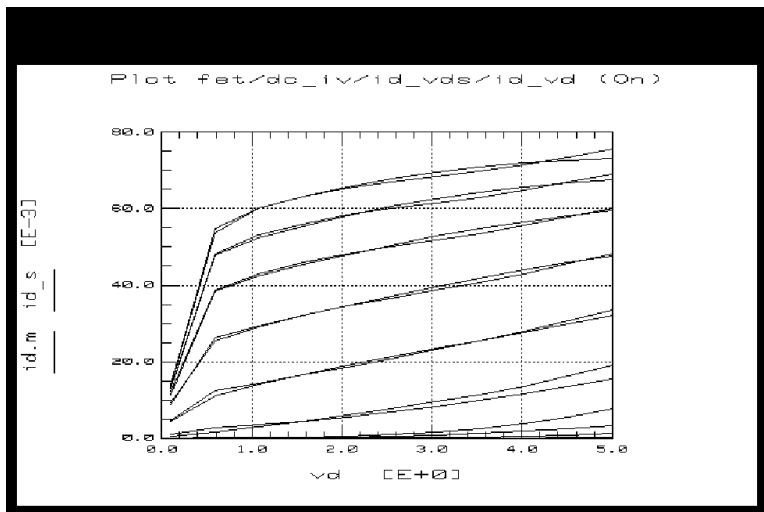
Figure 147   Example Plot of Final Optimized DC-IV Data

If you decide to rework the DC-IV extraction phase, make sure to reset EEFET3.peff to a high value such as 100.

5 Close the plots.

Extracting Charge and Dispersion Parameters

Charge parameters are divided into two groups:

- The gate charge parameter extraction is based on the S-parameter data measured at Vdso in *AC_at_VDSO/ Meas_Spars.*

- The output charge and dispersion parameter extractions are based on the S-parameter data measured with active DC bias in *AC_all/Meas_Spars.*

It is important that the individual extractions are based on the correct set of S-parameter data: the data from the same DUTs in which the extractions are grouped. If the setups are mixed within DUTs or performed in the wrong sequence, the results will be wrong.

A transform in each procedure computes a set of linear equivalent circuit parameters based on the measured S-parameter data. The actual model parameters are extracted from this linear equivalent circuit data.

The dispersion parameters are grouped here together with the charge parameters because they also are extracted from the S-parameter data measured with active bias.

Extracting Gate Charge Parameters

This procedure uses the S-parameter data you measured in the first Meas_Spars procedure. Since the model equations simplify at Vdso, the charge parameters C11o and C11th, Vinfl, and Deltgs can most accurately be extracted with Vds set to a constant value of Vdso. These are initial values for C11o and C11th.

Extracting the Linear Equivalent Circuit

1 From the AC_at_VDSO DUT, select the **Meas_Spars** setup and **Display Plots**.

2 From Extract/Optimize folder, select **mdl** and **Execute**. This computes and saves a set of linear equivalent circuit parameters based on the measured S-parameters. Observe the changes in the plots.

3 Close the plots.

Extracting C11 Capacitance with C11_vgs

This procedure extracts the C11 capacitance looking into the gate terminal.

1 Locate the values for EEFET3.c11o, c11th, vinfl, and deltgs (near the bottom of the table).

2 Select the setup **C11_vgs**.

3 From the Plots folder, select the **C11** plot, and **Display Plot**.

4 From the Extract/Optimize folder, select the **initialize** transform, and **Execute** to establish initial values.

5 Select the transform **opt_to_C11** and **Execute** to calculate the final values. You will see the traces converge on the plot, and the calculated values for the parameters will be entered in the Parameters table. The following figure is a typical plot of the input capacitance of a GaAs FET over all Vgs values at Vdso.
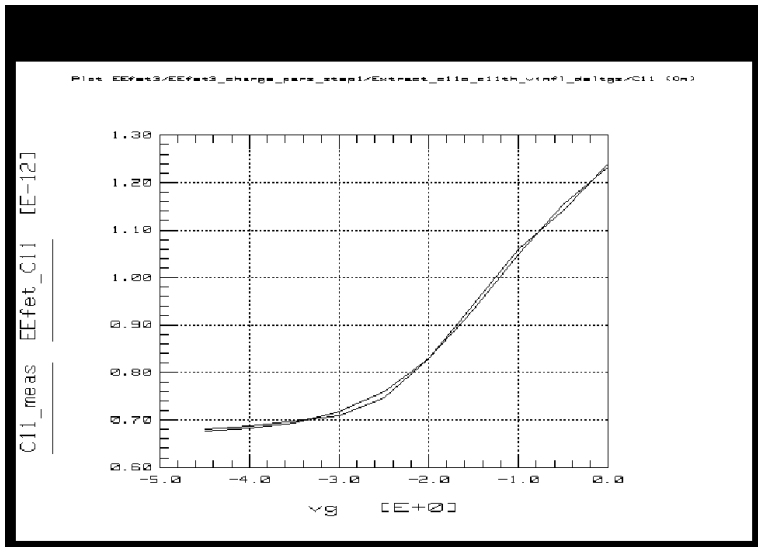
Figure 148   Example Plot of Gate Input Capacitance Over Vgs at Vdso

Generally you will not need to change any of the optimization limits for these gate charge parameters. However, the limits are very important to the correct functionality of the optimizer.

Extracting Gm and Gds with ac_gm_gds

This is an initial extraction for Gm and Gds. These parameter values will be refined in a later procedure.

1 Locate the values for EEFET3.gmmaxac, gammaac, kappaac, peffac, and vtoac in the Parameters table.

2 Select the setup **ac_gm_gds**, and **Display Plots**.

3 From Extract/Optimize folder, select **Init_dispersion** and **Execute**.

4 Select **opt_gm_fet3** or **opt_gm_hemt1**, and **Execute**. Observe the changes in the plots and the Parameters table values. You probably will not need to change the optimization limits.

There may be a slight tradeoff in match between Gm and Gds. When the Gds data is well-matched the Gm data may degrade. You may like to perform the optimization again to see if it improves the match of both Gm and Gds. Or you can repeat both the Init_dispersion transform and the opt_gm transform.

Note that Gm AC is lower than Gm DC because of trapping effects: this is not accounted for in most industry standard models.

5 Close the plots.

Extracting Output Charge Parameters

The extraction procedures in the following setups use the S-parameter data you measured in the second Meas_Spars procedure. It is essential you use that S-parameter data, and not jump directly to these extractions using the wrong set of S-parameter data.

These procedures start by extracting the C21 and C22 capacitances, and then fit Cdso and Tau to the extracted C21/C22 data using optimization. The final procedure extracts the dispersion parameters.

Extracting the Linear Equivalent Circuit

1 From the AC_all DUT, select the **Meas_Spars** setup and **Display Plots**.

2 From the Extract/Optimize folder, select **mdl** and **Execute**. This computes and saves a set of linear equivalent circuit parameters based on the S-parameters measured with bias. You will see the traces change on the plots, though a large amount of data is displayed and it is difficult to distinguish the convergence of individual traces.

3 Close the plots.

Extracting Initial Capacitance with Cgd_vgs_vds

This procedure extracts an initial value for the gate-to-drain capacitance.

1 Locate the values for deltds, lambda, and c12sat in the Parameters table.

2 Select the setup **Cgd_vgs_vds**.

3 From the Plots folder, select the **Cgd** plot, and **Display Plot**.

4 From the Extract/Optimize folder, select the transform **opt_to_Cgd** and **Execute**. Observe the changes in the plot and in the Parameters table values.

Generally you will not need to change any of the optimization limits for these charge parameters. However, the limits are very important to the correct functionality of the optimizer.

5 Close the plot.

Extracting Final Capacitance with Qg_vds_vgs

This procedures extracts final values for the capacitance parameters.

1 Locate the values for EEFET3.c11o and c11th in the Parameters table.

2 Select the setup **Qg_vds_vgs**.

3 From the Plots folder, select the **C11** plot, and **Display Plot**.

4 From the Extract/Optimize folder, select the transform **opt_C11_Cgd** and **Execute**. You will see the traces converge on the plot, and the parameter values may change in the table. You probably will not need to change any of the optimization limits. The following figure illustrates a typical C11 plot.
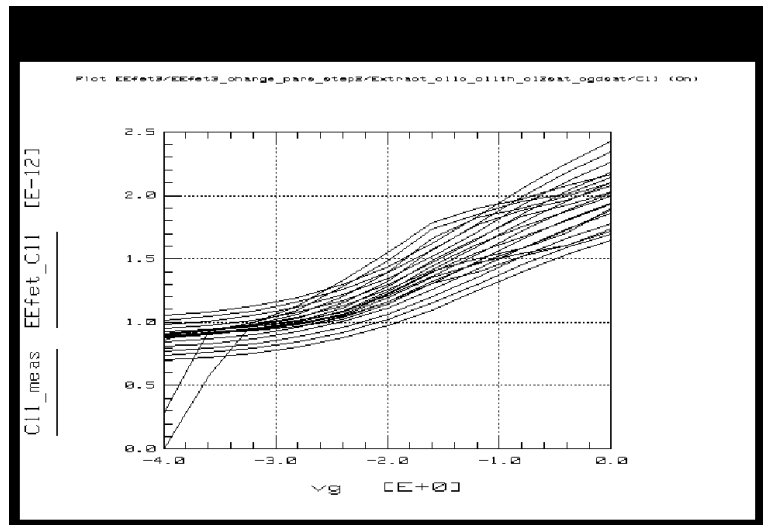
Figure 149   Example C11 Device Input Capacitance Plot

The C11 plot shows how the input capacitance values trend upward vs. Vgs from threshold, spreading as they go toward the upper limits.

5 Note the values for EEFET3.c12sat and cgdsat in the Parameters table.

6 Close the plot.

Extracting Tau and Ri with Tau_Ri

The extractions for Tau and Ri are grouped together, even though they are independent, because both parameters are dependent on good extraction of the extrinsic and package parasitics. Tau is the time constant delay of S21. It is hard to separate from the transcapacitance Cds, the output capacitance with a change in input voltage.

1 Locate the values for EEFET3.tau and cdso in the Parameters table.

2 Select the setup **Tau_Ri_Cds**.

3 From the Plots folder, select the **Tau** and **Cds** plots, and **Display Plot**.

4 From the Extract/Optimize folder, select the **opt_Tau** transform and **Execute**. This optimizes both Tau and Cds. Observe the data change on the trace, and possible parameter value changes for EEFET.tau and cdso. The following figure illustrates an example of plotted Tau data. The spread in data is caused by capacitance effects.
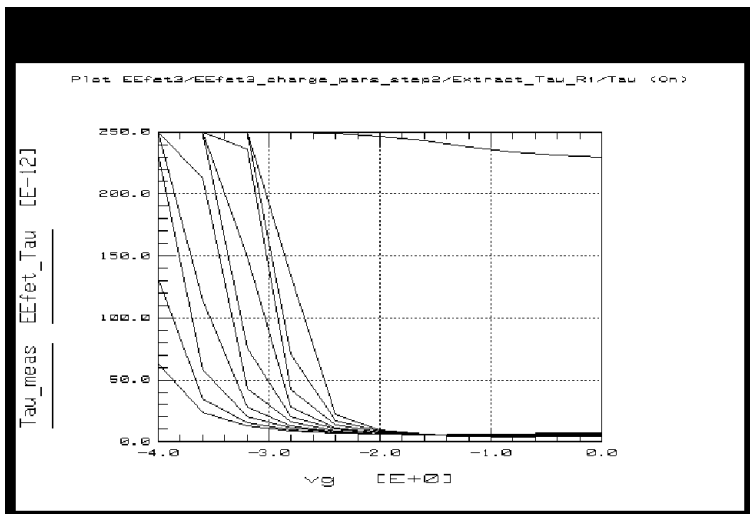
Figure 150   Example Tau Plot

If the Tau extraction yields a parameter value of zero or less, the problem may be in the earlier parasitic extractions. If the parasitic extractions look good and the Tau extraction looks bad, set a value for Tau manually. A good nominal value is 5.0 picoseconds for a 0.5 um GaAs FET, scaled approximately linearly with gate length for other FETs.

5 Close the Tau and Cds plots. Display the Ri plot.

6 Locate the value for EEFET3.ris in the Parameters table.

7 Select the **opt_ri** transform and **Execute**. Observe the data change on the trace, and a possible parameter value change for EEFET3.ris. The following figure illustrates example Ri data.
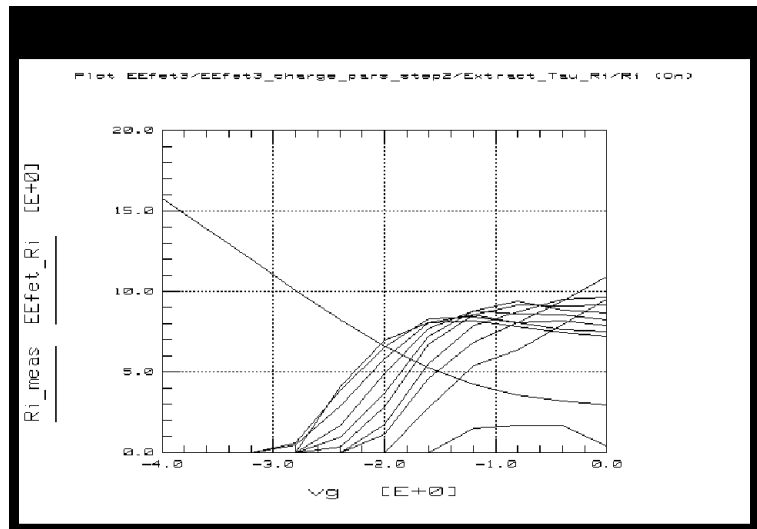
Figure 151   Example Ri Plot

If the Ri extraction yields a value of zero or less, it indicates that the contact resistance Rg was extracted too high. If this is the case, you may wish to re-iterate the rg_rd extraction for more accurate results. Ri is typically a value of only a few ohms, which may erroneously be attributed to Rg. The effect may be unimportant if the total of the gate resistances is

good. If the fit of total gate resistance to S11 is sufficiently good the iteration of the rg_rd contact resistance extraction is not necessary.

You can also reset the value of EEFET3.rg in the Parameters table a little lower, forcing the remaining resistance to be attributed to Ri. If you do this, or if you re-iterate the rg_rd contact resistance extraction, you *must* perform the mdl transform in the second Meas_Spars setup again before performing the opt_ri transform in *this* setup. Then check the Parameters table values for Ri again.

8 Close the plot.

Extracting Dispersion Parameters

Both Gm and Gds change in GaAs devices due to traps and other effects. This procedure adjusts the dispersion parameters that model this behavior. The setup used in this extraction is tied to the S-parameter measurement and linear equivalent model extracted in the second Meas_Spars setup. The mdl transform in that setup must be active (that is, no other S-parameters must be measured and no other mdl transform must be run since the one in the second Meas_Spars setup).

1 If the mdl transform in the second Meas_Spars setup has not been run, open that setup and **Execute** the transform.

2 Scroll the Parameters table to show some of the following parameters: EEFET3.gmmaxac, gammaac, kappaac, peffac, vdeltac, gdbm, kdb, and vdsm. You will not be able to display them all at the same time.

3 Select the setup **ac_gm_gds**.

4 From the Plots folder, select **Gm** plot, and **Display Plot**.

5 From the Extract/Optimize folder, select the transform **opt_gm_fet3** or **opt_gm_hemt1**, and **Execute**. Observe the traces converge on the plot, and the parameter values change in the table. The following figure illustrates example Gm data. Some dynamic thermal effects may be observed.
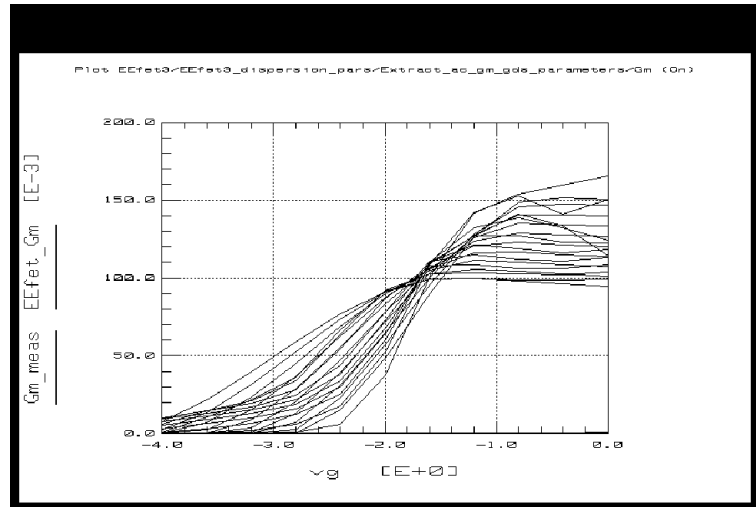
Figure 152   Example Gm Plot in the ac_gm_gds Setup

You can use the X-low/X-high function if you want to optimize over only a portion of the trace.

6   Scroll the Parameters table to show more of the relevant parameters.

7   Display the Gds plot.

8   Select the **opt_gds** transform, and **Execute**. Observe the traces converge on the plot, and the parameter values change in the table. The following figure illustrates example Gds data.
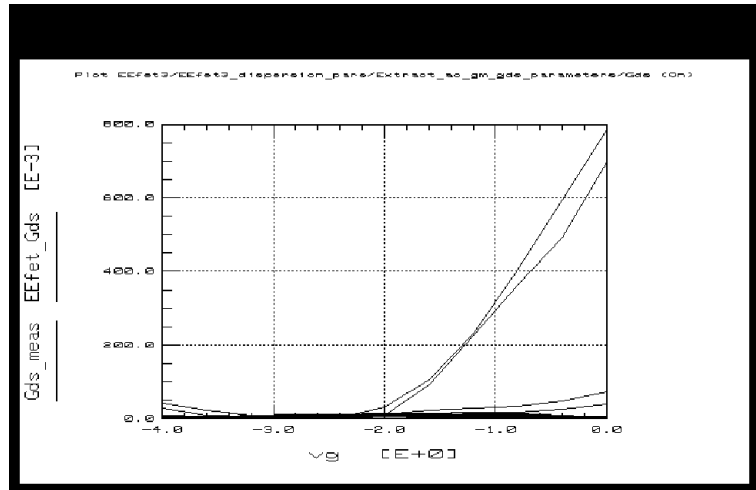
Figure 153   Example Gds Plot in the ac_gm_gds Setup

In some cases there may be a tradeoff between a good data fit at threshold and in the active region of operation.

Verifying the Model Against the Measured Data

This procedure uses the File_Validate setup to verify that modeled S-parameter data is comparable to measured data. An S-parameter measurement is performed at one bias value–the specific bias of expected device operation. This allows a single bias condition to be isolated for comparison purposes, which cannot be done with the multiple-bias conditions of earlier S-parameter measurements.

Defining the Instrument States

To define the DC Source/Monitor instrument state, follow these instructions:

- "Defining the DC Source/Monitor Instrument State" on page 595 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

•   "Defining the DC Source/Monitor Instrument State" on
    page 621 (for an Agilent 8753- or 8720-based system) in
    Appendix C, "Using IC-CAP with an Agilent 85123A Device
    Modeling System"

Use the same DC source/monitor and network analyzer settings
as you did for the Meas_Spars and other S-parameter
measurements. Then return to this chapter and continue.

The Input Variables

1 From the Utilities DUT, select the **File_Validate** setup and
observe the inputs. The following figure illustrates the inputs
used in this measurement.
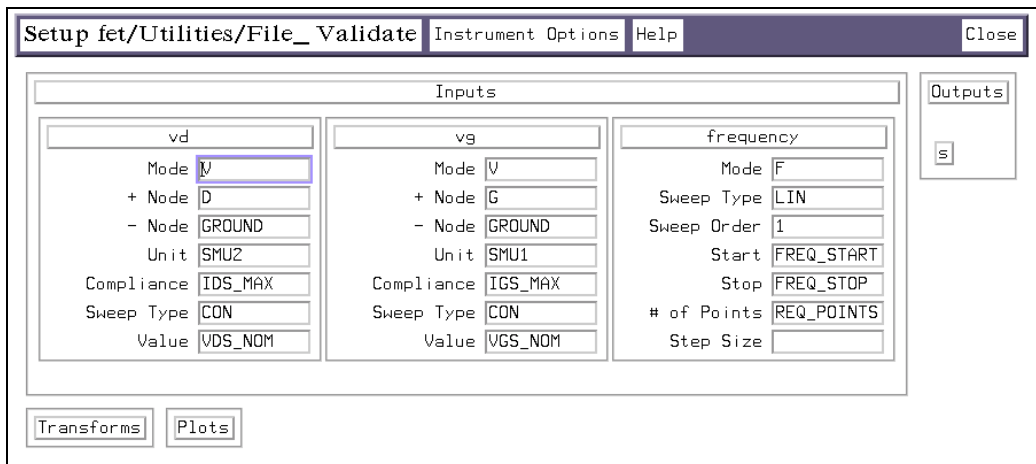
Figure 154   Inputs for Verification Data

•   The frequency inputs are set the same as for the other
    S-parameter measurements.
•   The gate voltage is set for a CONstant value of VGS_NOM,
    the value of Vgs that gives an Ids value representative of
    actual device operation.
•   The drain voltage is set for a CONstant value equal to the
    intended Vds of device operation.

Measuring and Plotting

1 With the File_Validate setup selected, select **Display Plots**, then click **Measure**.

2 From the Extract/Optimize folder, select **mdl** and **Execute**. This embeds the extrinsic elements and makes the model parameters comparable with the measured results.

3 Observe the measured and modeled S-parameter data on the plots. The plot below illustrates typically well-converged data.
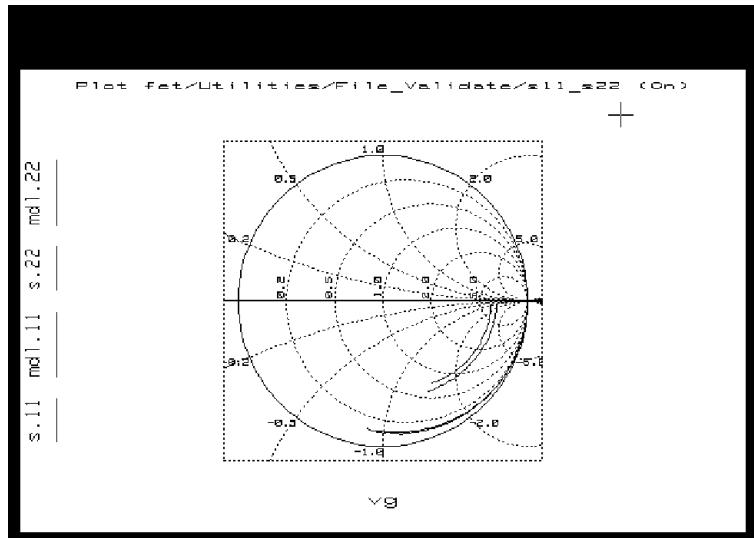
Figure 155   Measured and Modeled S11 and S22 Data

Storing the Model

This procedure stores the model in IC-CAP, including all the device-specific measured and simulated data. Once you have developed the device model, it can be used as a component in the Agilent RF and Microwave Design System (MDS). You can use it for circuit simulation, inserting it into circuits under design to emulate the characteristics of your device.

If you will be modeling large numbers of devices, it is worthwhile to organize your directory structure according to device type, geometry, and batch or wafer.

1 Decide where in your directory structure you intend to store this model file.

2 From the IC-CAP/Main window select **Model**, then select **File > Save As** from the pulldown menu. A dialog box is displayed.

3 Select a new filename that associates this model with the device, for example *eefet6_9jun95.mdl*. Type the full directory pathname and the filename into the dialog box.

4 Select **OK** or press Return to store the data and close the dialog box.

Saving the Parameters for Series IV Compatibility

A transform in the File_Validate setup lets you save the model parameters in a format that can be used by the Series IV Design System.

1 From the DUT/Setup panel select the **File_Validate** setup.

2 From the Extract/Optimize folder, select the transform **SaveFile**, and **Execute**.

3 Type a filename for the parameters and click **OK**. The parameters will be saved in a Series IV-compatible format. The format is MDIF (measurement data interchange format). When the transform is run, a copy of the MDIF file is written to the UNIX standard out window.

Saving the Parameters in SPICE Format

You can also save the list of individual model parameters in SPICE format for later access. These parameters can be read directly into MDS.

1 From the Model window choose **File > Export Data > Extracted Deck**. Select a new filename, for example *fet6pars.mdl*. Type the full pathname and your filename into the dialog box.

2 Click **OK** or press Enter to store the parameters in SPICE format.

This completes the example high-frequency EEfet3 measurement and parameter extraction procedure. For an explanation of how to use IC-CAP model data in MDS, refer to Appendix D, "Using IC-CAP Data with MDS."

References

1 End Resistance Technique

New Method to Measure the Source and Drain Resistance of the GaAs MESFET. Long Yang and Stephen I. Long. IEEE Electron Device Letters Vol. EDL-7, No. 2, February 1986.

2 Contact Parasitic Resistance and Inductance Method

A New Method for Determining the FET Small-Signal Equivalent Circuit. Gilles Dambrine, Alain Cappy, Frederic Heliodore, and Edouard Playez. IEEE Transactions on Microwave Theory and Techniques, Vol. 36, No. 7, July 1988.

3 Extraction of Package Parasitic and Intrinsic FET Equivalent Circuit Values

Direct Extraction of GaAs MESFET Intrinsic Element and Parasitic Inductance Values. Eric Arnold, Michael Golio, Monte Miller, and Bill Beckwith. IEEE MTT-S Digest pp. 359-362, 1990.
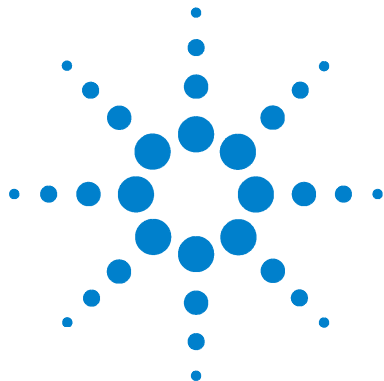
7

The Agilent EEBJT2 Model

The Agilent EEBJT2 model is a nonlinear bias-dependent packaged semiconductor device model for silicon BJTs, based on the well-known Gummel-Poon model, with modifications.

The model has been created for automatic parameter extraction from measured data including DC and S-parameter measurements. It can also be used for non-automated extraction or fine tuning of individual parameters. EEBJT2 provides an accurate reproduction of transistor behavior, with DC bias, bias-dependent S-parameters including effects of package parasitics, and true nonlinear harmonic output power. The model is quasi-static, analytical, and isothermal. The model does not scale with area, since parameters are extracted directly from measured data and not from layout considerations.

Agilent Technologies

Getting Started in IC-CAP

The starting point of this tutorial assumes the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"
  - Wiltron 360-based system—Refer to your Wiltron documentation
- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.
- The IC-CAP software has been properly installed on the computer.
- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed as follows, based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Rename a medium-power SMU connected to the device base as SMU1

Rename a high-power SMU connected to the collector as SMU2

Rename the GNDU unit (which must be connected to the emitter) as GND

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on launching the IC-CAP software and opening the model file you need. The EEBJT2 model file is under:

/examples/model_files/bjt/HPEEbjt2.mdl

Then return to this chapter and continue with the procedures here.

Before you continue with the IC-CAP procedures, make a new directory in an appropriate location in the UNIX structure, using the mkdir command, to store the data and model files that will be generated. Give the directory a name that will associate it with the devices you are going to model. Use the cd command to change to the new directory in UNIX.

Using the EEBJT2 Model

The base-emitter current in the EEBJT2 model has been changed from the Gummel-Poon and other earlier models. These models assume that the non-leakage base-emitter current is related to the collector-emitter current by a simple constant known as beta. In EEBJT2, the non-leakage base and collector currents are modeled with separate equations. This approach simplifies the extraction process and, in some cases, may provide a superior fit to the measured data. For more detail refer to the IC-CAP *Reference* manual.

The EEBJT2 model provides the following features:

- High-level accuracy for a wide range of processes and operating conditions.

- Extraction of package parasitics, significant at higher frequencies.

- A complete set of model parameters that can be acquired directly from measurements (without knowledge of process information).

- A high level of automation.

The DUTs and Setups

The DUT/Setup panel is part of the Model window, which is the central access point for all the measurement and parameter extraction procedures you perform. The Model window includes tabs to access the macros, the Model Parameters table, and other IC-CAP features. To open this model, click the **Examples** button on the toolbar and locate */model_files/bjt/HPEEbjt2.mdl*.
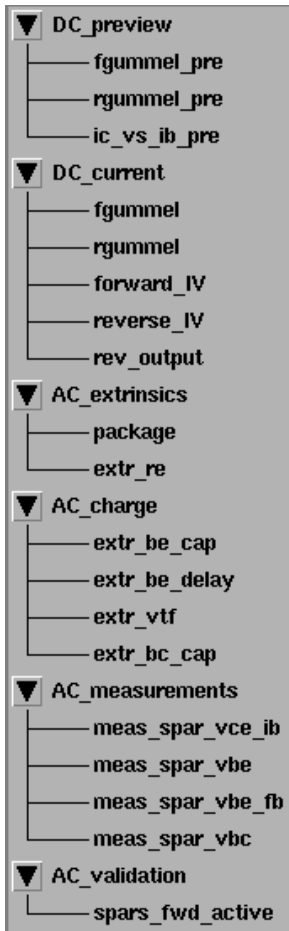
Figure 156   Agilent EEBJT2 DUT/Setup Panel

In the DUT/Setup panel, the DUTs (for example DC_preview)—indicated by the inverted triangles—are groupings of similar measurement setups used to extract related model parameters. The setups (for example fgummel_pre) contain the information used to define the inputs and outputs for each measurement, as well as their measurement hardware configurations and their associated transforms and plots.

The measurement and parameter extractions are most easily performed with macros. If you use macros, you will not need to use the individual setups. However, if you want to extract individual parameters or "fine-tune" a parameter, you will use the individual setups.

README is not actually a macro, but an abbreviated version of the instructions in this chapter. It is most appropriate for users who are somewhat familiar with the model and with IC-CAP. You can view the README file in a separate window by clicking **Detach**. Then resize the window and move it away from the central work area. Scroll the window ahead as you work through the model.

The EEBJT2 model parameters, with their descriptions and default values are shown the following table.

Table 43   EEBJT2 Model Parameters

Parameter	Description	Default (Units)
NF	Forward-current emission coefficient.	1.2 (unitless)
VAF	Forward early voltage.	30.0 (V)
IKF	Corner for forward beta high-current roll-off.	100 (A)
ISE	B-E leakage saturation current.	1.0e–20 (A)
NE	B-E leakage emission coefficient.	1.2 (unitless)
NR	Reverse-current emission coefficient.	1.2 (unitless)
VAR	Reverse early voltage.	3.0 (V)
IKR	Corner for reverse beta high-current roll-off.	100 (A)
ISC	B-C leakage saturation current.	1.0e–20 (A)
NC	B-C leakage emission coefficient.	1.2 (unitless)
RB	Base resistance.	1.0 (ohms)
RE	Emitter resistance.	1.0 (ohms)
RC	Collector resistance.	1.0e–6 (ohms)

Table 43   EEBJT2 Model Parameters (continued)

Parameter	Description	Default (Units)
CJE	B-E zero-bias depletion capacitance (CJE, VJE, and MJE determine nonlinear depletion-layer capacitance for B-E junction).	1.0e–12 (F)
VJE	B-E junction built-in potential (CJE, VJE, and MJE determine nonlinear depletion-layer capacitance for B-E junction).	0.75 (V)
MJE	B-E junction exponential factor (CJE, VJE, and MJE determine nonlinear depletion-layer capacitance for B-E junction).	0.33 (unitless)
TF	Ideal forward transit time (TR and TF, along with the depletion-layer capacitances, model base charge storage effects; TF may be bias-dependent).	0.0 (seconds)
XTF	Coefficient of bias-dependence for TF.	0.0 (unitless)
VTF	Voltage dependence of TF on B-C voltage.	0.02 (V)
ITF	Parameter for high-current effect on TF.	0.0 (A)
CJC	B-C zero-bias depletion capacitance (CJC, VJC, and MJC determine nonlinear depletion-layer capacitance for B-C junction).	1.0e–12 (F)
VJC	B-C junction built-in potential (CJC, VJC, and MJC determine nonlinear depletion-layer capacitance for B-C junction).	0.75 (V)
MJC	B-C junction exponential factor (CJC, VJC, and MJC determine nonlinear depletion-layer capacitance for B-C junction).	0.33 (unitless)
TR	Ideal reverse transit time (TR and TF, along with the depletion-layer capacitances, model base charge storage effects).	0.0 (seconds)
FC	Forward-bias depletion capacitance coefficient.	0.5 (unitless)
TYPE	Model type--NPN (1) or PNP (2).	npn (unitless)
ISF	Forward saturation current.	1.0e–20 (A)

Table 43 EEBJT2 Model Parameters (continued)

Parameter	Description	Default (Units)
IBIF	Forward base saturation current.	1.0e–20 (A)
NBF	Forward base emission coefficient.	1.2 (unitless)
ISR	Reverse saturation current.	1.0e–20 (A)
IBIR	Reverse base saturation current.	1.0e–20 (A)
NBR	Reverse base emission coefficient.	1.2 (unitless)
TAMB	Ambient temperature of measurement and model parameter extraction.	25 (degrees C)
LB	Base inductance.	0.0 (H)
LC	Collector inductance.	0.0 (H)
LE	Emitter inductance.	0.0 (H)
CXBE	External base-emitter fringing capacitance.	0.0 (F)
CXBC	External base-collector fringing capacitance.	0.0 (F)
CXCE	External collector-emitter fringing capacitance.	0.0 (F)

Setting the Model Variables

Before you begin the device measurements, it is necessary to set numerical values for some of the model variables. Most of these variables are set in macros, either automatically or with the aid of explanatory prompts. Other variables must be set manually, before any macros are run. All variable values are stored in the model variable table. They are referenced by mnemonic name from the various macros and/or setups throughout the model. Using mnemonics eliminates the need to set numerical input values for every measurement setup.

NOTE	Some variables have fixed values and must not be changed (refer to "Variables with Fixed Values" on page 408).

Entering the DC Supply Resolution Limits

The first set of variables defines the resolution limits of the DC current/voltage supply (for example an Agilent 4142B DC source/monitor).   These variables are set manually, and must be set before any macros are run. When you set start, stop, and number of points values for a measurement sweep, the software automatic-ally generates the step size. If the step size is beyond the resolution of the DC supply, an error message is displayed. Presetting these resolution limits can prevent error messages from occurring. The resolution limit variables (which are near the end of the variable table) are:

Table 44

IB_RES	Current resolution of SMU connected to input port.
IC_RES	Current resolution of SMU connected to output port.
VB_RES	Voltage resolution of SMU connected to input port.
VC_RES	Voltage resolution of SMU connected to output port.

Appropriate values for these variables can be found in the documentation for the DC supply (for example Agilent 4142B).

Setting Circuit Solver Variables

The transforms that perform a DC analysis include a simple circuit solver that implements the EEBJT2 equations directly into the transforms. You will normally not see this circuit nor need to use it. However, there may be conditions when you would like to exercise control over the convergence criteria of the solver. The next three variables make this possible:

Table 45

NTRIALS	Maximum number of iteration trials used by the Newton-Raphson algorithm.
TOL_VOLT	The algorithm will exit when the sum of the magnitudes of the voltage corrections drops below this value.

Table 45

TOL_FUNC	The algorithm will exit when the sum of the magnitudes of the function corrections drops below this value. The algorithm solves for the internal voltages of the device, but tries to match the model's predicted responses to the independent port currents or voltages being measured. If voltages are being set on both ports, the algorithm tries to zero out the functions v1_meas – v1_mod and v2_meas – v2_mod.

Variables with Fixed Values

The following variables are preset and should remain fixed. Do not change them:

Table 46

PATH_LOCAL	This is used by some transforms to look in the local path for data.
INFINITY	Preset to the value 1.0e50. It is used in some transforms to represent a very large number.
SIMULATOR	This should always have the value mns. During the model extraction and verification, the simulation is performed without the benefit of an external simulator. However, MNS can be used for additional verification of device behavior.

Setting Variables in Macros

The remaining variables are set in the variable initialization macros—the first three macros in the list after README:

- Data_sheet_vars
- Misc_sweep_vars
- Pre_meas_vars

These macros should be run before any measurements or extractions are performed (except that Pre_meas_vars makes the DC_preview measurements and sets variables based on the results). Perform these initialization macros in the order listed. Select the required macro and **Execute**. Respond to the prompts.

Data_sheet_vars Macro

This macro is used to set the following variables:

Table 47

VEB_MAX	Maximum emitter-base voltage.
VCB_MAX	Maximum collector-base voltage.
VCE_MAX	Maximum collector-emitter voltage.
IC_MAX	Maximum collector current.
PT_MAX	Maximum total device power dissipation.

Values for these variables are commonly available on data sheets of packaged parts. These variables are critical to the data acquisition in many setups, and they must be set before measurement of any device.

Misc_sweep_vars Macro

This macro sets many of the variable values for different sweeps used throughout the EEBJT2 model. Most of the default values for variables set by this macro are suitable for most devices, so you need only run this macro if you want to change any default values. Otherwise continue directly to the Pre_meas_vars macro. The following variables are included:

Table 48

FGML_START	Forward Gummel sweep start voltage.
FGML_PTS	Number of data points taken in forward Gummel sweep.
RGML_START	Reverse Gummel sweep start voltage.
RGML_PTS	Number of data points taken in reverse Gummel sweep.
IB_PRE_PTS	Number of data points taken in Ic vs. Ib preview measurement.
IB_PTS	Number of base current traces in forward and reverse IV measurement.

Table 48

VCE_START	Forward IV measurement Vce start value.
VCE_PTS	Number of Vce points in forward IV measurement.
VCE_REV_START	Reverse IV measurement Vce start value.
VCE_REV_PTS	Number of Vce points in reverse IV measurement.
VCE_START_AC	Starting value for Vce in forward S-parameter measurements.
VCE_PTS_AC	Number of Vce points in forward S-parameter measurements.
VBE_CAP_PTS	Number of Vbe bias points for BE junction S-parameter measurements.
VBE_FB_PTS	Number of Vbe bias points for forward-bias BE junction S-parameter measurements.
VBC_CAP_PTS	Number of Vbc bias points for BC junction S-parameter measurements.
VJ_ON	Approximate junction turn-on voltage.

This macro requests the ambient temperature of the lab where the device is being measured. (This is actually the parameter TAMB, rather than a measurement variable. It is listed with its default value in Table 43.)

Pre_meas_vars Macro

NOTE   Do not run the initialization macros if you have existing measured data, as the data will be overwritten.

This macro should be run after the Data_sheet_vars macro and before performing any of the measurements in the DC_current DUT. You can also run the Misc_sweep_vars macro before this macro if you wish to change any of the default values in it. The Pre_meas_vars macro makes the DC_preview measurements

and sets variables based on the results. Therefore you must have your test equipment set up and switched on and your device ready to measure.

This macro provides a straightforward way to interactively set the following variables:

FGML _STOP	Forward Gummel sweep stop voltage.
RGML_STOP	Reverse Gummel sweep stop voltage.
VCE_REV_STOP	Reverse IV measurement Vce stop value.
VCE_STOP	Forward IV measurement Vce stop value.
IC_COMP	Collector current compliance limit.
IB_START	Forward and reverse IV measurement base current start value.
IB_STOP	Forward and reverse IV measurement base current stop value.

While the macro is executing, it displays the following plots:

- Forward Gummel
- Reverse Gummel
- Base current vs. collector current at VCE_STOP

Use the plots to visually determine appropriate values for the upper limit sweep variables FGML_STOP, RGML_STOP, and IB_STOP respectively. The objective is to set the variables as high as possible without violating the collector current compliance. If the compliance limit is violated, select "n" as indicated in the prompt to re-measure, and re-enter a new (lower) sweep limit variable.

You can now recall the same set of device variables any time you wish to measure and model similar devices. As you input more variables into the table in later procedures (particularly the macros) you will probably want to save the variable table again, overwriting this version.

Calibrating the Network Analyzer

All the EEBJT2 S-parameter measurements require only one calibration: a swept broadband calibration across the operating frequency range of the device under test. Good calibration of the network analyzer is critical to a good measurement and extraction.

For step-by-step calibration procedures, refer to:

- Agilent 8510-based system—"Calibrating the Network Analyzer" on page 589 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- Agilent 8753-based system—"Calibrating the Network Analyzer" on page 631 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

- Wiltron 360 network analyzer—Refer to the Wiltron documentation

Make the following additional setting changes:

- Set the network analyzer START frequency to the lowest possible frequency in the range of the network analyzer and bias networks: low frequencies allow more accurate AC conductances to be extracted.

- Set the STOP frequency to the upper limit of the frequency range for the device under test. Frequencies up to Ft are useful but not required. Good package and charge extractions require frequencies high enough to see roll-off and package effects clearly.

- Set the NUMBER of POINTS to 51 or 101 to provide clear and usable S-parameter plots: more points only add more time and memory requirement to the measurement and extraction.

Remember these frequency values: you will also need to enter them in IC-CAP when instructed. If you do not set these values correspondingly, the calibration will be invalid.

All S-parameter measurements for the EEBJT2 model are performed over the same frequency range. However, some of the extractions are performed at a single frequency point, for example in the Extract_extrin macro. These extractions must be

performed at a calibrated frequency point. Only the measured frequency points in the sweep are calibrated, that is, points equal to the start frequency plus an integer multiple of the step size.

In the Extract_extrin macro, a transform takes any arbitrary frequencies you may have entered, and moves to the nearest calibrated frequency point.

NOTE	If you set a MHz value in IC-CAP, write *MEG* in full. IC-CAP is not case-sensitive and assumes that *m* or *M* corresponds to *milli*. If an error message is displayed during the measurement that indicates a network analyzer frequency value is too low, you may have entered *M* only.

Remember the number of the network analyzer cal set or register where you store the calibration. You will need to use this number in the instrument options of *each* S-parameter measurement when you define the instrument states.

Using Macros for Measurements and Extractions

Macros are the easiest and fastest way to perform a complete set of measurement and extraction procedures. Macros automate the modeling procedure so that a series of similar devices can be measured and modeled quite quickly with minimal user intervention. As you perform the extraction macros, the default values in the Model Parameters table are replaced by extracted values.

If individual parameters need to be extracted, or if you want to "fine-tune" a particular parameter, you can use the individual setups in the Model window instead of the macros. They are explained later in the chapter.

The following table describes the EEBJT2 macros and lists the setups whose transforms are used to accomplish the same thing if you want or need to do it manually.

Table 49 EEBJT2 Macros

Run This Macro	To Do This	Or Do It Manually By
Extract_DC	Measure, extract, and optimize DC parameters	Executing the transforms in the following setups (DC_current DUT): fgummel rgummel forward_IV reverse_IV rev_output
AC_Measure	Measure S-parameters	Executing the transforms in the following setups (AC_extrinsics DUT): package and (AC_measurements): meas_spar_vce_ib meas_spar_vbe meas_spar_vbe_fb meas_spar_vbc
Extract_extrin	Extract the parasitic inductances as well as the base and emitter resistances	Executing the transforms in the following setups (AC_extrinsics DUT): package extr_re
Extract_junc_cap	Extract the reverse-bias depletion capacitance parameters for both the BE and BC junctions	Executing the transforms in the following setups (AC_charge DUT): extr_be_cap extr_bc_cap
Extract_transit	Extracts the transit time related parameters for the BE junction	Executing the transforms in the following setups (AC_charge DUT): extr_be_delay extr_vtf

DC Measurements and Extractions

The Extract_DC macro performs the same DC measurements as the setups in the DC_current DUT in the Model window. It then extracts the DC parameters from the measured data and enters them in the Parameters table. The DC measurements should

always be performed first, and the DC parameters must be the first parameters extracted. Later extractions are dependent on a good DC extraction.

To run the DC measurement and extraction macro:

1 Click the **Macros** tab.

2 Select **Extract_DC > Execute**.

The macro displays the DC plots one at a time. As each plot appears, one or more dialog boxes appear prompting you to input data values. You need to have some knowledge of the device to input appropriate values. When the macro is complete, the results of the extraction are printed in the Status window, and the corresponding values are listed in the Parameters table.

S-Parameter Measurements

NOTE
The network analyzer must be calibrated before you perform any S-parameter measurements; if you have not yet done so, refer to the section, "Calibrating the Network Analyzer" on page 412.

The AC_Measure macro performs an S-parameter measurement equivalent to the one in the package setup (AC_extrinsics DUT). It is a swept-frequency S-parameter measurement at a single DC bias point. The macro displays prompts for you to enter the frequency start and stop values and number of measurement points, and the Ib and Vce bias points. It then performs the measurement and plots the results.

- If you are not satisfied with the measurement results, you can exit the macro and recalibrate the network analyzer.

- If you are satisfied with the measurement results, you can have the instrument states (options) copied to the other S-parameter measurement setups (in the AC_measurements DUT).

NOTE

You can enter additional bias variables and the rest of the S-parameter measurements are performed automatically. You should do this if you are going to perform the AC extraction macros, because the Extract_junc_cap and Extract_transit macros use the data from these additional S-parameter measurements. These are swept frequency measurements with swept DC bias.

AC Extractions

The AC extractions require running three separate macros:

- Extract_extrin
- Extract_junc_cap
- Extract_transit

These macros do not make measurements, but use the S-parameter data collected during the execution of the AC_measure macro.

Extract_extrin Macro: Parasitic Extractions

This macro extracts the parasitic inductances as well as the base and emitter resistances. It is thus equivalent to the package and extr_re setups in the AC_extrinsics DUT. It prompts you for extraction frequencies for conductances, capacitances, and inductances. If you use arbitrary frequency values, a transform in the macro moves the extraction to the nearest calibrated frequency point. Calibrated frequency points are any points in the calibrated network analyzer frequency range that are equal to the start frequency plus an integer multiple of the step size. The macro extracts the parasitic inductances and the base and emitter resistances, and lists their values in the Parameters table. It extracts re by optimizing the modeled intrinsic gm to fit the measured intrinsic gm.

Extract_junc_cap Macro: Junction Capacitance Extractions

This macro extracts the reverse-bias depletion capacitance parameters for both the BE and BC junctions. It is thus equivalent to the extr_be_cap and extr_bc_cap setups under AC_charge. It first displays plots of cbe_vs_vbe and cbc_vs_vbc,

then performs optimizations and lists the new parameter values in the Parameters table. The parameters optimized are cje, vje, mje, cjc, vjc, and mjc. No measurements are taken in this macro: be sure to run the AC_measure macro first.

Extract_transit: Transit Time Related Parameter Extractions

This macro is equivalent to the extr_be_delay and extr_vtf setups in the AC_charge DUT. It displays plots of cbe_vs_ic and ft_vs_ic. It prompts you for a value for the parameter xtf and optimizes tf and itf. It then displays the cbe_vce plot from the extr_vtf setup, and extracts vtf. It enters the new parameter values in the Parameters table. No measurements are taken in this macro: be sure to run the AC_measure macro first.

Executing Individual Measurement Transforms

Though the macros are the fastest and easiest way to perform the measurement and extraction procedures, you may sometimes want to extract parameters individually, or you may want to "fine-tune" a particular parameter.

The rest of this chapter explains in detail how to make preview device measurements and set model variables from the results obtained; how to make the measurements required for specific extractions; how to perform the individual extractions and optimizations; and how to save the resulting model.

Note that, among the individual setups, the fgummel and rgummel setups each include a macro transform to automate the measurement and extraction procedures for those particular setups.

Defining the Instrument States

For each of the measurement setups, it is necessary to define the instrument states (options) for the measurement instruments used in that setup. The instrument states need to be defined individually for each setup. With minor exceptions, the options must be the same for all setups that use the same instruments.

To set the instrument options for any given setup:

1 Select the setup of interest.

2 Click the **Instrument Options** tab and the display changes to show the options for the instruments used in that measurement.

   • In the DC setups, the options for the DC source/monitor are displayed.

   • In the S-parameter setups, the options for both the DC source/monitor and the network analyzer are displayed.

The instrument state settings will be explained in this chapter with their corresponding setups.

Making DC Measurements

The easiest and fastest way to make all of the DC measurements (and perform the DC extractions) is to use the Extract_DC macro, after first running the variable initialization macros Data_sheet_vars, Misc_sweep_vars, and Pre_meas_vars. These initialization macros set most of the variables needed for the DC measurements. Additional variables are set in the Extract_DC macro or the individual setups.

Measuring with the DC_preview DUT

NOTE	Although measurement instructions are provided here for each setup in the DC_preview DUT, it is easier to use the Pre_meas_vars macro to automate the DC preview measurements.

The DC_preview DUT setups are used to obtain a good set of measured DC-IV curves on which most of the other measurements will be based. These setups are:

• fgummel_pre

• rgummel_pre

• ic_vs_ib_pre

Each of the three setups is designed to give you a quick preview of the measurement and, based on the results, set the related variables that control the final measurement.

Previewing forward Gummel with fgummel_pre

This setup is used to preview the forward Gummel measurements for both the base and collector currents.

To define the DC Source/Monitor instrument state, follow these instructions:

- Agilent 8510-based system—"Defining the DC Source/Monitor Instrument State" on page 595 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- Agilent 8753-based system—"Defining the DC Source/Monitor Instrument State" on page 621 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

These procedures configure IC-CAP with the instrument state settings (instrument options) for the Agilent 4142B DC source/monitor. Then return to this chapter and continue.

The input values define the device configuration, the sweep values that will be used to make the measurement, and the compliance limits for the power supply.

To view the fgummel_pre input values:

Select the **fgummel_pre** setup in the DC_preview DUT.

- You will not need to change the input values, although you may need to change some variable values as a result of the measurement. However, the inputs show you what measurement is being made and how the measurement variables are used.

- Notice that Mode is set to V for all inputs, to provide a voltage at all the device terminals.

- The Unit names must conform to the names you set in the hardware window (see "Getting Started in IC-CAP" on page 400). They must also correspond with the actual plug-in SMU connections from the Agilent 4142. The vb unit must be set to SMU1 and the vc unit to SMU2. The ve unit must be set to GND.

- Compliance values limit SMU output voltage or current and prevent damage to the device under test, as well as to the SMUs, bias networks, and probes if used. With a voltage input, compliance refers to current. In this model, the compliance values are preset. Note that the compliance at the collector is set for maximum collector current.

- Notice the input vb is set for a linear voltage sweep from start to stop values preset with variables.   The start value is the approximate junction turn-on voltage. It may be necessary to change the variable value of FGML_STOP after the measurement.

- Notice the input ve is grounded. All measurements in this model are made with the BJT in a common-emitter configuration.

- Notice the input vc is set for a sweep synchronized to the vb sweep.

- The outputs are ib (base current) and ic (collector current).

- The output Type is M for measured. The other possibilities are S for simulated and B for both.

NOTE   In measuring chip devices, remove any high-intensity light sources such as microscope light before taking a measurement. This is particularly important if currents in the nanoamp range are being measured. One simple method of blocking light is to place an opaque plastic box on the fixture directly above the device.

The measurement process triggers the device measurement, controlling the DC source/monitor to supply the signals defined in the instrument options window and the inputs, and monitors the results.

To view the sample fwd_Gummel plot:

1 From the Plots folder, select the **fwd_Gummel** plot, and click **Display Plot**.
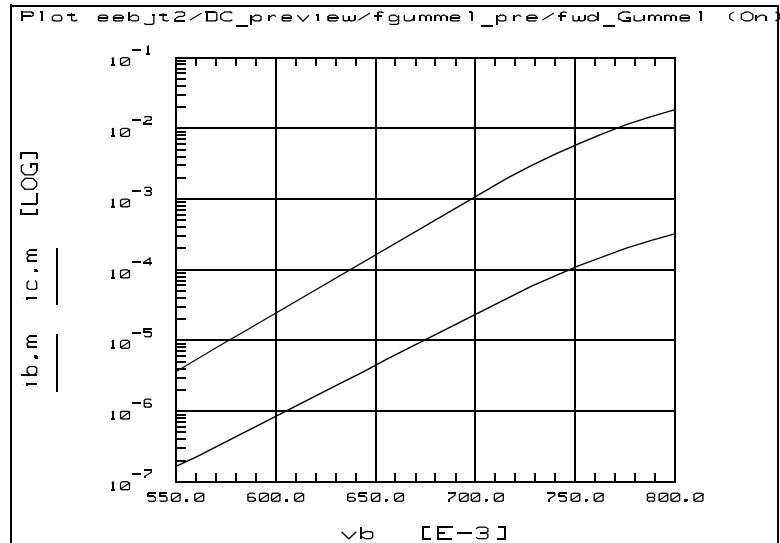
Figure 157   Example fgummel_pre Data Plot

- The lower curve is base current and the upper curve is collector current.

- The stop value for the base voltage sweep has been preset to a value large enough to drive the device into compliance on the collector port to obtain the maximum amount of data possible within the compliance range. However, it is not required that you drive the device into compliance, so you can use a lower value of FGML_STOP if you prefer.

- The recommended approach is to drive the device into compliance using the preset FGML_STOP value, then find the maximum value of vb below compliance. Set FGML_STOP to this value. To do this, go to the variable table and find the variable FGML_STOP, then manually change its value in the table. Then re-measure and make sure the new value does not violate the compliance limit.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Previewing Reverse Gummel with rgummel_pre

The rgummel_pre setup is used to preview the reverse Gummel measurements for both the base and emitter currents.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings.

To view the rgummel_pre input values:

Select the **rgummel_pre** setup.

• Note that in this setup, vb is set to a constant value of 0.0.

• The input vc is set for a linear voltage sweep from start to stop values preset with variables. Note that the start value is −VJ_ON for this reverse measurement   It may be necessary to change the variable value of RGML_STOP after the measurement.

• The compliance at the collector is set for maximum collector current, as in the fgummel_pre measurement. The stop value for the collector voltage sweep has been preset to a value large enough to drive the device into compliance on the emitter port.

To view the sample rev_Gummel plot:

1 From the Plots folder, select the **rev_Gummel** plot, and click **Display Plot**.
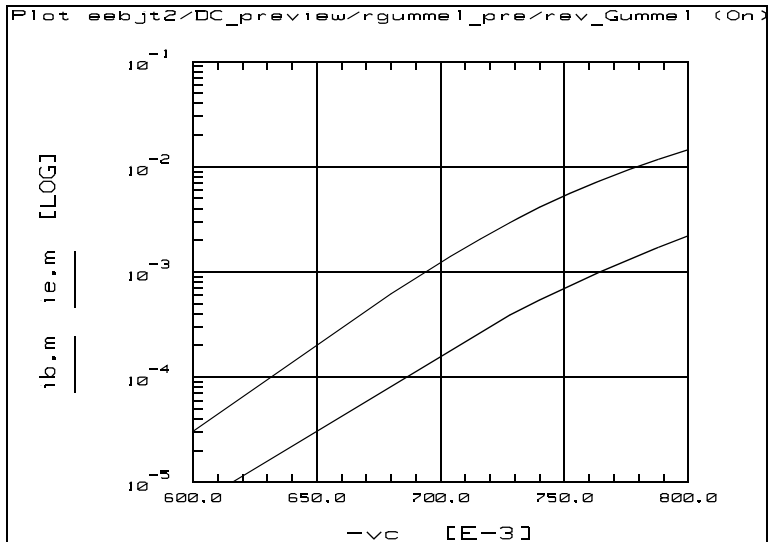
Figure 158   Example rgummel_pre Data Plot

- The lower curve is the base current and the upper curve is the emitter current.
- The stop value of the collector voltage sweep has been preset to a value large enough to drive the device into compliance on the emitter port, to obtain the maximum amount of data within compliance.
- Drive the device into compliance using the preset RGML_STOP value, then find the maximum value of vc below compliance. Set RGML_STOP to this value, and perform the measurement again.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Previewing Collector Current with ic_vs_ib_pre

The ic_vs_ib_pre setup is used to preview the collector current at VCE_STOP (the stop value for Vce used in the forward_IV measurement). The objective of this setup is to find the largest value of IB_STOP such that the collector current compliance is not violated. Note that it is easier to run the Pre_meas_vars macro to automate the DC preview.

To define the DC source/monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings.

To view the ic_vs_ib_pre input values:

Select the **ic_vs_ib_pre** setup.

• In this setup, the input on the base is current, therefore the ib Mode is set to I. With this current input, the compliance is a voltage.

• The input ib is set to sweep from 0.0 to IB_STOP, the base current stop value used for the forward_IV and reverse_IV measurements. vc is set to a constant value of VCE_STOP.

• You will need to set the values of VCE_STOP and IC_COMP manually in the variable table. Set VCE_STOP to a value <= VCE_MAX. Set IC_COMP to the lesser of IC_MAX or PT_MAX/VCE_STOP.

• You can also set the values for IB_STOP and IB_PRE_PTS in the variable table, but the default values are usually adequate for most devices.

To view the sample ic_vs_ib plot:

1 From the Plots folder, select the **ic_vs_ib** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change. If you are not satisfied with the results, change the value of *IB_STOP* in the variable table and perform the measurement again.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Measuring with the DC_current DUT

NOTE	Do not run the initialization macros if you have existing data in this model file, or the data will be overwritten. Do not run the Extract_DC macro if you already have acceptable measured DC data.

The DC_current DUT makes five DC measurements using the following setups:

- fgummel
- rgummel
- forward_IV
- reverse_IV
- rev_output

To prevent possible bias oscillation, put the network analyzer in hold sweep mode.

Measuring with the fgummel Setup

This setup measures the operating range of base and collector currents with the transistor in forward active mode. It measures Ic and Ib with respect to Vbe.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings except set Integ Time to L (long) to slow the measurement and reduce noise at the lower end of the curve.

To view the fgummel input values:

From the DC_current DUT, select the **fgummel** setup.

- The inputs are almost the same as for the fgummel_pre setup, except that now the vb Start value is set to FGML_START, the forward Gummel sweep start voltage, which you can set in the variable table if you wish to make it different from the junction turn-on voltage.

To view the sample fwd_Gummel plot:

1 From the Plots folder, select the **fwd_Gummel** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Measuring with the rgummel Setup

The rgummel setup measures the base and emitter currents with the transistor in the reverse active mode. It measures Ib and Ie with respect to –Vce.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings except set Integ Time to L (long).

To view the rgummel input values:

Select the **rgummel** setup.

• The inputs are almost the same as for the rgummel_pre setup, except that now the vc Start value is set to RGML_START, the reverse Gummel sweep start voltage, which you can set in the variable table if you wish to make it different from the inverse value of the junction turn-on voltage.

To view the sample rev_Gummel plot:

1 From the Plots folder, select the **rev_Gummel** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Measuring with forward_IV

The forward_IV setup measures Ic with respect to Vce at several values of Ib.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings.

To view the forward_IV input values:

Select the **forward_IV** setup.

- The vc values are set to sweep the collector voltage across the operating range of the device.
- The ib values are set to measure several values of base current. IB_PTS is set for the number of base current traces.

To view the sample ic_vce plot:

1 From the Plots folder, select the **ic_vce** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change. If you are not satisfied with your results, you can change the values of variables in the variable table. For example, if the Ib curves are too low or too high, you can change the IB_START or IB_STOP values. If you do change any variable values, perform the measurement again.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Measuring with reverse_IV

The reverse_IV setup measures Ie with respect to Vce at several values of Ib. The Vce values are chosen to measure the device in the reverse active mode.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings.

To view the reverse_IV input values:

Select the **reverse_IV** setup.

- The ib values are set the same as for the forward_IV measurement. This setup measures the same number of base current traces as the forward_IV.

- The vc sweep variables are set to negative voltage values.

To view the sample ie_vce plot:

1 From the Plots folder, select the **ie_vce** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change. If you are not satisfied with your results, you can change the variable table values, as in the forward_IV measurement, and perform the measurement again. Do not change the number of measurement points for either ib or vc

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.pre*.

Measuring with rev_output

The rev_output setup measures Ie with respect to Vec, with the device in the reverse active mode. The measured data will be used later in the procedures to extract the reverse early voltage parameter EEBJT2.var. It is important that the Gummel measurements are performed first.

To define the DC Source/Monitor instrument state:

Follow the same procedure from the fgummel_pre setup, using the same settings except set Integ Time to L (long) to reduce noise, because one of the transforms uses a numerical derivative, and the extraction will be inaccurate if the data is noisy.

To view the rev_output input values:

Select the **rev_output** setup.

- You will need to set the variable value for VBC_EARLY manually before you make the measurement. VBC_EARLY is the Vbc value at which an Ie vs. Vec trace is measured. Scroll the variable table until you find VBC_EARLY, and enter a value. Select a value in the linear region of the reverse Gummel plot, near the high end of the region but below the series resistance drop effects.

- The default value for EARLY_PTS is generally acceptable.

To view the sample ie_vec plot:

1 From the Plots folder, select the **ie_vec** plot and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change. If you are not satisfied with your results, you can change the value of VBC_EARLY in the variable table, and perform the measurement again.

3 To save the plotted data, select **File > Save As** and a filename with the suffix *.set*.

Making S-parameter Measurements

The AC_measure macro performs all the S-parameter measurements in the model, including copying the instrument states and frequency range from the first S-parameter measurement setup to all the others. But first you must run the variable initialization macros Data_sheet_vars, Misc_sweep_vars, and Pre_meas_vars.

NOTE   Do not run the initialization macros if you have existing data in this model file, or the data will be overwritten. Do not run the AC_measure macro if you already have acceptable measured S-parameter data.

These initialization macros set most of the variables needed for the S-parameter measurements; a few additional variables are set in the set_bias transform macros in each S-parameter setup. If you intend to measure the S-parameters in one setup only, you will first need to run the set_bias transform macro in that

setup. The macro prompts you for bias values specific to the measurement. You will also need to set the sweep parameters manually (once for all the measurements).

NOTE   The network analyzer must be calibrated before you perform any S-parameter measurements. Also make sure the network analyzer is in continuous sweep mode. For details see "Calibrating the Network Analyzer" on page 412.

Measuring with the AC_extrinsics DUT

The AC_extrinsics DUT contains two setups:

- package—Contains measurement and extraction transforms. Measuring is described here; the extraction is described later in the extraction section.
- extr_re—Contains extraction transforms only and will be described later in the extraction section.

Measuring with package

The package setup measures S-parameters in the typical operating region of the device, with constant values of base and collector bias. The data measured here will be used to extract the extrinsic parasitic base resistance and the extrinsic parasitic inductances.

Defining the Instrument States

You will need to set the instrument states for both the DC source/monitor and the network analyzer. For details, refer to:

- "Instrument States for Swept Measurements" on page 598 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
- "Instrument States for Swept Frequency Measurements" on page 624 (Agilent 8753- or 8720-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

NOTE

For the DC source/monitor, use the same settings as you did for the DC measurements. Calibrate the network analyzer. In the instrument options for the network analyzer, be sure to set the Cal Set No to the cal set or register where you save your calibration.

You will use these instrument states in all the setups that make S-parameter measurements with the network analyzer.

To view the package input values:

Select the **package** setup.

• Unless you used the AC_measure macro, you need to set the frequency sweep variables manually. Scroll the variable table to show the values for *FREQ_START*, *FREQ_STOP*, and *FREQ_PTS*. Manually enter values for these variables that correspond with the sweep values used for the network analyzer calibration.

• If you wish, you can also set the bias sweep variables *IB_TYP* and *VCE_TYP* manually, or you can use the set_bias transform macro.

• To do so, select **Extract/optimize > set_bias > Execute**. Enter typical base current and collector voltage values close to the intended point of operation.

To view the sample S-parameter plots:

1 From the Plots folder, select each plot of interest and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As**.

Measuring with the AC_measurements DUT

The setups in the AC_measurements DUT are used to obtain S-parameter data for extractions performed in other setups, as shown in the following table.

Table 50   S-parameter Measurement and Extraction setups

DUT/Setup for Measurement	DUT/Setup for Corresponding Extraction
AC_measurements/meas_spar_vce_ib	AC_extrinsics/extr_re AC_charge/extr_vtf
AC_measurements/meas_spar_vbe	AC_charge/extr_be_cap
AC_measurements/meas_spar_vbe_fb	AC_charge/extr_be_delay
AC_measurements/meas_spar_vbc	AC_charge/extr_bc_cap

NOTE   The package setup (in the AC_extrinsics DUT) is also used for making AC measurements, but unlike those listed in Table 50, the package setup also contains the related extraction transforms.

The extractions listed in Table 50 will work only if the corresponding AC_measurement setup contains previously measured S-parameters. Each of the extraction setups must have bias input variables identical to the input variables of the corresponding AC_measurement setup. It is important that they remain the same: do not change the inputs.

The transform s_mod in each S-parameter measurement setup computes the modeled S-parameters of the small-signal equivalent circuit for comparison with the measured results.

Measuring with meas_spar_vce_ib

The meas_spar_vce_ib setup measures the device S-parameters in the forward active region with Ib and Vce as the control variables.

To define the instrument states:

Follow the same procedure you used in the package setup. Use the same settings for both the DC source/monitor and the network analyzer. (The instrument states need to be set independently for each setup, unless you ran the AC_measure macro.)

To view the meas_spar_vce_ib input values:

Select the **meas_spar_vce_ib** setup.

- The frequency variables *FREQ_START*, *FREQ_STOP*, and *FREQ_PTS* must have values in the variable table that correspond with the frequency values used in calibrating the network analyzer.

- The base current is set to sweep over the same range as in the forward_IV DC measurement.

- The vc values are set to sweep the collector voltage across the approximate operating range of the device.

- You can set the VCE_STOP_AC value using the set_bias transform macro. Select **Extract/Optimize > set_bias > Execute**, and respond to the prompts.

To view the sample S-parameter plots:

1 From the Plots folder, select each plot of interest and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As**.

Measuring with meas_spar_vbe

The meas_spar_vbe setup measures S-parameters with Vbc equal to zero and Vbe varying from reverse bias to forward bias. This will be used for extraction of parameters associated with the BE junction capacitance.

To define the instrument states:

Follow the same procedure you used in the package setup. Use the same settings.

To view the meas_spar_vbe input values:

Select the **meas_spar_vbe** setup.

- The frequency inputs are the same as for the meas_spar_vce_ib setup, and are also the same in the other S-parameter setups.

- The input vc is set for a sweep synchronized to the vb sweep.
- The vb values are set to vary the device performance from reverse bias to forward bias. VBE_CAP_PTS is a default value.
- You can set the vb start and stop values for the measurement using the set_bias transform macro. Select **Extract/Optimize > set_bias > Execute**, and respond to the prompts.

To view the sample S-parameter plots:

1 From the Plots folder, select each plot of interest and click **Display Plot**.
2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.
3 To save the plotted data, select **File > Save As**.

Measuring with meas_spar_vbe_fb

The meas_spar_vbe_fb setup measures S-parameters with Vbc equal to zero and Vbe forward biased. This measured data will also be used for extraction of parameters associated with the BE junction capacitance.

To define the instrument states:

Follow the same procedure you used in the package setup. Use the same settings.

To view the meas_spar_vbe_fb input values:

Select the **meas_spar_vbe_fb** setup.

- The frequency inputs are the same as in all the other S-parameter setups.
- The input vc is set for a sweep synchronized to the vb sweep.
- The vb values are set to forward bias the device. VBE_FB_PTS is a default value.
- You can set the vb start and stop values for the measurement using the set_bias transform macro. Select **Extract/Optimize > set_bias > Execute**, and respond to the prompts.

To view the sample S-parameter plots:

1 From the Plots folder, select each plot of interest and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As**.

Measuring with meas_spar_vbc

The meas_spar_vbc setup measures S-parameters, with Vbe equal to zero and Vbc varying from reverse bias to forward bias.

To define the instrument states:

Follow the same procedure you used in the package setup. Use the same settings.

To view the meas_spar_vbc input values:

Select the **meas_spar_vbc** setup.

• The frequency inputs are the same as in all the other S-parameter setups.

• The vb inputs set the base voltage to zero.

• The vc input values are set to vary the device performance from reverse bias to forward bias. VBC_CAP_PTS is a default value.

• You can set the vc start and stop values for the measurement using the set_bias transform macro. Select **Extract/Optimize > set_bias > Execute**, and follow the prompts.

To view the sample S-parameter plots:

1 From the Plots folder, select each plot of interest and click **Display Plot**.

2 To initiate the measurement process, from the Measure/Simulate folder, click **Measure** and observe the plot(s) change.

3 To save the plotted data, select **File > Save As**.

Executing Individual Extraction Transforms

NOTE   The fastest and easiest way to perform the extractions is to use the macros provided. Or you can perform each extraction step individually to allow complete process control in case you want to extract parameters individually or "fine-tune" a particular parameter. This section explains how to extract parameters using the individual transforms; for details on using the macros, refer to "Using Macros for Measurements and Extractions" on page 413.

When the measurements are finished and you have a complete set of measured data for a particular device, you can begin the extractions. If you saved the measured data, you can perform an extraction later, at any time. This also offers the opportunity for the measurement and extraction to be done by different people.

Extracting with the DC_current DUT

This section describes performing extractions by executing individual transforms associated with the DC_current DUT.

Extracting parameters with the fgummel_macro transform

The easiest way to perform the extraction of all the forward Gummel parameters is to use the fgummel_macro transform. This extracts a number of critical parameters from the data measured earlier. If you are not using the macros under the Macros tab to automate the extractions, it is important that this is the first extraction performed, because effective extractions in later setups are dependent on accurate parameters obtained in the fgummel and rgummel setups.

1 Select the **fgummel** setup.

2 Click **Extract/Optimize** and select **fgummel_macro > Execute**.

   • The macro displays the measured data on the fwd_Gummel plot.

   • Screen prompts ask you to input limit values for the extraction.

- Observe the simulated traces converge to the measured traces on the plot as the macro optimizes the parameters.

- The Parameters table is updated with the new optimized forward Gummel parameters.

| NOTE |

A similar method is available in the rgummel setup, where the rgummel_macro transform is the easiest way to perform the extraction of all the reverse Gummel parameters.

Extracting Parameters with Individual fgummel Transforms

This is the method to use if you need to extract individual parameters or "fine-tune" a particular parameter. Use the Parameters table, scrolling as necessary, to see the new extracted values of the parameters. Because this setup extracts a fairly large number of parameters, the procedure is more complex than for many setups. The procedures presented in this section—for each transform—assume the fgummel setup is the active setup.

To compute hfe from the measured data:

From the Extract/Optimize folder, select **hfe_fwd_meas > Execute**. To look at the hfe trace, select **Plots > hfe_vs_ic > Display Plot**.

After setting the variables on which the extractions and optimizations depend, you can extract and optimize the base current parameters EEBJT2.ibif, nbf, ise, and ne.

To set the variables:

1 Select **Plots > fwd_Gummel > Display Plot**.

2 To view a linear portion of the fgummel traces, click the **Setup Variables** tab. The setup variable table appears.

3 From the plotted data, choose a linear portion of the collector current trace, excluding any noisy data at the low end. Click the mouse pointer at the lower and upper boundary points of the linear portion to make a box on the trace, as shown in Figure 159.

4 From the Plot window, select **Options > Copy to Variables**. This changes the X_LOW and X_HIGH values for this plot to the boundary points you selected.

5 In the setup variable table, set VBE_LOW_TR to a value less than X_HIGH, where the slope of the base current trace is still the same as at X_HIGH.
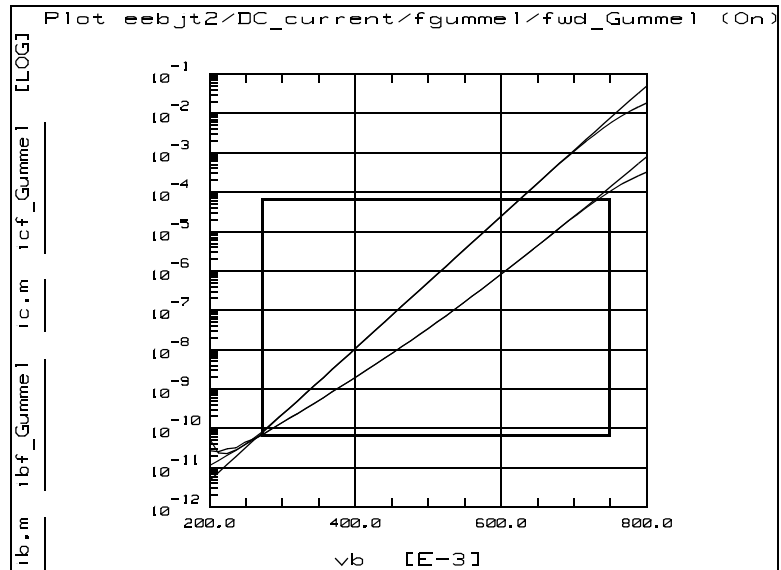
Figure 159 Selecting a Linear Portion of the Ic Trace

6 In the setup variable table, set VBE_HIGH_LK to a value greater than X_LOW where the slope of the base current trace is the same as at X_LOW.

 • The region between VBE_LOW_TR and X_HIGH is the transport region where the parameters ibif and nbf can be estimated.

 • The region between X_LOW and VBE_HIGH_LK encloses the leakage region where the parameters ise and ne can be estimated.

Once these variables are set, the remaining transforms can be executed. Many of the transforms rely on these variables, therefore the extractions will not work properly unless the variables are first set correctly.

To execute the remaining transforms in the fgummel setup:

1 Scroll the Parameters table to display isf and nf.

2 Select **Extract/Optimize**.

3 Select **extr_isf_nf > Execute**. Observe the changes in the Parameters table values.

> **NOTE** The values extracted for the parameters isf and nf are a function of both the input data and the parameter EEBJT2.var. Therefore, after you extract var in the rev_output setup, you will need to return to this setup and execute the extr_isf_nf transform again. (Note that this is handled automatically in the Extract_DC macro.)

4 Select **icf_Gummel > Execute**. The transform computes the simulated forward Gummel collector current.

5 Scroll the Parameters table to display EEBJT2.ibif and nbf.

6 Select **est_ibif_nbf > Execute**. Observe the changes in the Parameters table values.

> **NOTE** This transform zeroes the parameter EEBJT2.ise so that the effects of EEBJT2.ibif and nbf are the only ones visible in simulations immediately following the extraction. This makes the quality of the fit easier to determine.

7 Select **ibf_Gummel > Execute**. The transform computes the simulated forward Gummel base current, so that the result of the ibif and nbf extraction is observable on the plot.

Now you must make a decision based on the results you have seen thus far:

- If the Gummel fit in the transport region between VBE_LOW_TR and X_HIGH looks good, you can stop the extraction at this point.

- If you want to extract the leakage parameters ise and ne, you can skip the next two transforms (set_opt_vars_fwd, Step 1 and opt_ibif_nbf, Step 2) and go directly to est_ise_ne (Step 3).

- If you are not satisfied with the Gummel fit in the transport region and you do not need to extract the leakage parameters, perform the next two transforms (set_opt_vars_fwd, Step 1 and opt_ibif_nbf, Step 2) and then stop.

1 Select **set_opt_vars_fwd > Execute**. This sets optimization limits for the ibif, nbf, ise, and ne parameters. The limits are set to a specific percentage above and below the parameter values. The variable *PAR_OPT_TOL* controls the percentage variation.

2 The optimization performed with the opt_ibif_nbf transform should only be used if the base current leakage parameters are to be neglected. Otherwise, use the opt_ise_ne_ibif_nbf transform. If you use the opt_ibif_nbf transform, be sure to execute the transform est_ibif_nbf first.

3 Scroll the Parameters table to display EEBJT2.ise and ne.

4 Select **est_ise_ne > Execute**. Observe the value changes in the Parameters table.

5 Select **ibf_Gummel > Execute** again so that you can view the results of the parameter estimates for ise and ne on the plot.

Check the fit of the Gummel plot again and proceed as follows:

- If the fit looks good, the extraction is done.

- If you think it is necessary to optimize the fit, execute the following two transforms (set_opt_vars_fwd, Step 1 and opt_ise_ne_ibif_nbf, Step 2).

1 Select **set_opt_vars_fwd > Execute**.

2 Select **opt_ise_ne_ibif_nbf > Execute**. This optimizes all the parameters ibif, nbf, ise, and ne. You can see the final values in the Parameters table.

Extracting Parameters with Individual rgummel Transforms

Use this method if you need to extract individual parameters or "fine-tune" a particular rgummel parameter. Use the Parameters table, scrolling as necessary, to see the new extracted values of the parameters. The steps presented in this section—for each transform—assume the rgummel setup is the active setup.

To compute the emitter current (ie):

1 Select the **rgummel** setup.

2 Select **Extract/Optimize**.

3 Select **ie > Execute**. The transform computes the emitter current from the measured base and collector currents.

After setting the variables on which the extractions and optimizations depend, you can extract and optimize the base current parameters EEBJT2.ibir, nbr, isc, and nc.

To set the variables:

1 Select **Plots > rev_Gummel > Display Plot**.

2 To view a linear portion of the rgummel traces, click the **Setup Variables** tab. The setup variable table appears.

3 From the plotted data, choose a linear portion of the emitter current trace, excluding any noisy data at the low voltage end. Click the mouse pointer at the lower and upper boundary points of the linear portion of the trace.

4 From the Plot window, select **Options > Copy to Variables** to change the X_LOW and X_HIGH values for this plot to the boundary points you selected.

5 In the setup variable table, set VBC_LOW_TR to a value less than X-HIGH, where the slope of the base current trace is still the same as at X_HIGH.

6 In the setup variable table, set VBC_HIGH_LK to a value greater than X_LOW where the slope of the base current trace is the same as at X_LOW.

  • The region between VBC_LOW_TR and X_HIGH is the transport region where the parameters ibir and nbr can be estimated.

- The region between X_LOW and VBC_HIGH_LK encloses the leakage region where the parameters isc and nc can be estimated.

Once these variables are set, the remaining transforms can be executed. Many of the transforms rely on these variables, therefore the extractions will not work properly unless the variables are first set correctly.

1 Scroll the Parameters table to display isr and nr.

2 Select **Extract/Optimize**.

3 Select **extr_isr_nr > Execute**. Observe the changes in the Parameters table values.

NOTE	The values extracted for the parameters isr and nr are a function of both the input data and the parameter EEBJT2.vaf. Therefore, after you extract vaf in the forward_IV setup, you will need to return to this setup and execute the extr_isr_nr transform again. (Note that this is handled automatically in the Extract_DC macro.)

---

4 Select **icr_Gummel > Execute**. The transform computes the simulated reverse Gummel collector current.

5 Scroll the Parameters table to display EEBJT2.ibir and nbr.

6 Select **est_ibir_nbr > Execute**. Observe the value changes in the Parameters table.

NOTE	This transform zeroes the parameter EEBJT2.isc so that the effects of EEBJT2.ibir and nbr are the only ones visible in simulations immediately following the extraction. This makes the quality of the fit easier to determine.

---

7 Select **ibr_Gummel > Execute**. The transform computes the simulated reverse Gummel base current, so that the result of the ibir and nbr extraction is observable on the plot.

Now you must make a decision based on the results you have seen thus far:

- If the Gummel fit in the transport region between VBC_LOW_TR and X_HIGH looks good, you can stop the extraction at this point.
- If you want to extract the leakage parameters isc and nc, you can skip the next two transforms (set_opt_vars_rev, Step 1 and opt_ibir_nbr, Step 2) and go directly to est_isc_nc (Step 3).
- If you are not satisfied with the Gummel fit in the transport region and you do not need to extract the leakage parameters, perform the next two transforms (set_opt_vars_rev, Step 1 and opt_ibir_nbr, Step 2) and then stop.

1 Select **set_opt_vars_rev > Execute**. This sets optimization limits for the ibir, nbr, isc, and nc parameters. The limits are set to a specific percentage above and below the parameter values. The variable PAR_OPT_TOL controls the percentage variation.

2 The optimization performed with opt_ibir_nbr should only be used if the base current leakage parameters are to be neglected. Otherwise, use the opt_isc_nc_ibir_nbr transform. If you use this opt_ibir_nbr transform, be sure to execute the transform est_ibir_nbr first.

3 Scroll the Parameters table to display EEBJT2.isc and nc.

4 Select **est_isc_nc > Execute**. Observe the value changes in the Parameters table.

5 Select **ibr_Gummel > Execute** again so that you can view the results of the parameter estimates for isc and nc on the plot.

Check the fit of the reverse Gummel plot again.

- If the fit looks good, the extraction is done.
- If you think it is necessary to optimize the fit, perform the following two transforms (set_opt_vars_rev, Step 1 and opt_isc_nc_ibir_nbr, Step 2).

1 Select **set_opt_vars_rev > Execute**.

2 Select **opt_isc_nc_ibir_nbr > Execute**. This optimizes all the parameters ibir, nbr, isc, and nc. You can see the final values in the Parameters table.

Extracting Parameters with forward_IV (DC)

The forward_IV setup is used to extract the parameters EEBJT2.vaf, ikf, and rc. It is important to understand that this extraction will be effective *only* if the Gummel parameters have been properly extracted first.

1 Select the **forward_IV** setup.

2 Select **Plots > ic_vce > Display Plot**.

3 Select **Extract/Optimize**.

4 Select **set_limits > Execute**. A series of dialog boxes appears, prompting you to input minimum and maximum Vce and Ic values for the optimization.

5 Select **ic_mod > Execute**. This computes the modeled (simulated) collector current using Ib and Vce as the independent variables.

6 Select **set_vaf_to_limit > Execute**. This sets the parameter EEBJT2.vaf to its upper optimization limit VAF_MAX. It is useful to perform this transform before opt_ikf, because a high value for vaf helps to give a good upper boundary to EEBJT2.ikf.

7 In the Parameters table, set EEBJT2.rc to 0.0 for the ikf extraction.

8 Select **opt_ikf > Execute**. This performs a random optimization on the parameter EEBJT2.ikf. When executed with EEBJT2.rc = 0.0 and EEBJT2.vaf set to a large value, this optimization gives a good upper bound for EEBJT2.ikf.

9 Select **set_ikf_max > Execute**. This sets the variable IKF_MAX equal to the parameter EEBJT2.ikf, for use in the macro Extract_DC.

10 Select **opt_vaf_ikf > Execute**. This optimizes the parameters EEBJT2.vaf and ikf to fit the measured forward IV collector current. This optimization should be used only after you obtain a reasonable estimate for an upper bound on ikf.

11 Select **opt_rc > Execute**. This performs a random optimization of EEBJT2.rc to the measured forward IV collector current. The primary purpose is to get an initial estimate of rc.

12 Select **set_opt_vars_rc > Execute**. This sets the optimization limit variables for EEBJT2.rc. The limits are set to a specific percentage above and below the current values of rc:. The variable PAR_OPT_TOL controls the percentage variation. The primary purpose of this transform is to set the variables to a reasonable window after executing the opt_rc optimization and before executing the final opt_vaf_ikf_rc optimization.

13 Select **opt_vaf_ikf_rc > Execute**. This optimizes all three parameters to fit the measured forward IV collector current. This transform should be used only after you use the previous extraction steps to find reasonable estimates for these three parameters. Usually windowing is not necessary, but you can use the X-LOW/X-HIGH and Y-LOW/Y-HIGH functions if breakdown or self-heating effects are visible at high values of Vce, to obtain a linear data trace for the optimization. You can see the final values in the Parameters table and observe the traces converge on the plot.

14 Now that you have extracted a final value for vaf, go back to the rgummel setup and select **extr_isr_nr > Execute** again.

Extracting Parameters with reverse_IV (DC)

The reverse_IV setup is included for validation and reference only; it does not include any extractions.

1 Select the **reverse_IV** setup.

2 Select **Plots > ie_vce > Display Plot**.

3 Select **Extract/Optimize**.

4 Select **ie_meas > Execute**. The transform displays the measured data on the plot.

5 Select **ie_mod > Execute** to display the modeled data.

6 Observe the plot to compare the modeled data to the measured data.

Extracting Parameters with rev_output (DC)

The rev_output setup is used to extract and validate the reverse early voltage parameter EEBJT2.var. This extraction will be effective *only* if the Gummel parameters have been properly extracted first.

1 Select the **rev_output** setup.

2 Select **Plots > ie_vec > Display Plot**.

3 Select **Extract/Optimize**.

4 Select **ie > Execute**. The transform computes the measured emitter current from the measured base and collector currents.

5 Select **ie_mod > Execute**. This computes the modeled (simulated) emitter current.

6 Select **go > Execute**. This computes the derivative of the emitter current with respect to the voltage Vec. Scroll the Parameters table to display EEBJT2.var.

7 Select **est_var > Execute**. This computes an approximation of EEBJT2.var from a closed-form equation, using a numerical derivative based on the measured data to estimate var. The method is described in the book "Modeling the Bipolar Transistor," by Ian Getreu (see "Modeling the Bipolar Transistor. Ian Getreu. 1976 Tektronix Inc." on page 455). This step should be done before the optimization.

8 Select **opt_var > Execute**. This optimizes EEBJT2.var to fit the modeled ie data to the measured ie. You can observe the traces converge on the plot, and the final value for var in the Parameters table.

9 Now that you have extracted a final value for var, go back to the fgummel setup and select **extr_isf_nf > Execute** again.

Extracting with the AC_extrinsics DUT

The AC_extrinsics DUT contains two setups:

- package—Contains measurement and extraction transforms. The extraction is described here; measuring was described earlier in the measurement section.

- extr_re—Contains extraction transforms only and is described here in detail.

Extracting Parameters with package (AC)

The package setup extracts the extrinsic parasitic base resistance EEBJT2.rb and the extrinsic parasitic inductances lb, lc, and le.

1   Select the **package** setup. Select **Extract/Optimize**.

2   Select **set_freq_pars > Execute**. This transform looks for any arbitrary user-specified frequency values that are not calibrated, and resets them by moving them to the nearest calibrated frequency point. A calibrated frequency point is any point in the measured frequency sweep that is equal to the start frequency plus an integer multiple of the step size.

3   Select **s_mod_int > Execute**. This computes the intrinsic modeled S-parameters of the small-signal BJT equivalent circuit.

4   In the Parameters table find EEBJT2.lb, lc, and le. Set the values for these three inductance parameters to 0.000.

5   Select **estimate_ind > Execute**. This extracts the parasitic inductances from the measured data based on the method of Arnold et al, documented in "Direct Extraction of GaAs MESFET Intrinsic Element and Parasitic Inductance Values" (see "Modeling the Bipolar Transistor. Ian Getreu. 1976 Tektronix Inc." on page 455).

6   Open the delta_z_im plot and the S-parameter plots not suffixed _pre.

7   Select **s_mod_int > Execute** again. The s_mod and delta_z transforms are automatically recomputed and their results updated on the S-parameter plots and the delta_z_im plot. The modeled S-parameters should fit the measured data reasonably well after this step. Slight errors in the S11 fit will be accounted for in the next step. The traces in the delta_z_im plot should exhibit a linear dependence with frequency.

8   Locate EEBJT2.rb in the Parameters table.

9 Select **opt_rb > Execute**. This optimizes the parameter
EEBJT2.rb by minimizing the error between measured and
modeled S11. This also automatically calls the transform
s_mod, which computes the modeled extrinsic S-parameters
of the small-signal BJT equivalent circuit.

> **NOTE** The transform set_bias is actually a macro that was used in the
> measurement procedure for setting the sweep variables IB_TYP and
> VCE_TYP. Do not use it now.

Extracting Parameters with extr_re (AC)

This setup extracts the emitter resistance parameter
EEBJT2.re. It is important that all the DC_current extractions
are performed before this one.

This setup uses the measured S-parameter data from the
meas_spar_vce_ib setup, and will only work if that measured
S-parameter data exists. Note that this setup and the
meas_spar_vce_ib setup have identical sets of bias input
variables. It is important that they remain the same: do not
change the inputs.

1 Select the **extr_re** setup.

2 Select **Plots > gm_vs_vce > Display Plot**.

3 In the Parameters table find EEBJT2.re.

4 Select **Extract/Optimize**.

5 Select **sdta > Execute**. This reads the measured data,
de-embeds the extrinsics, and transforms the intrinsic data
into a table of bias-dependent small-signal element values.
This is the data on which the rest of the transforms are
performed.

6 Select **set_limits > Execute**. A series of dialog boxes appear
prompting for you to enter minimum and maximum Vce and
gm values for the gm optimization.

7 Select **gm_int_mod > Execute**. This computes the intrinsic
transconductance of the model.

8 Select **opt_re_random > Execute**. This uses a random
optimization to minimize the error between the measured

intrinsic gm and the modeled intrinsic gm, thus optimizing the emitter resistance re. Observe the re value change in the Parameters table.

9 Select **opt_re_grad > Execute**. This uses a gradient optimization to further minimize the error between the measured intrinsic gm and the modeled intrinsic gm.

Extracting with the AC_charge DUT

These setups are used for extractions only. For measured data, the AC_charge setups rely on the S-parameter measurements made in the setups of the AC_measurements DUT.

NOTE	If you do not use the Measure_AC macro to perform the measurements, you will need to set the variables *FREQ_START*, FREQ_STOP, and FREQ_PTS manually in the variable table. The values you select must correspond with the sweep values used for the network analyzer calibration.

Extracting Parameters with extr_be_cap (AC)

This setup extracts the base-emitter junction depletion capacitance parameters EEBJT2.cje, vje, and mje. It uses the measured S-parameter data from the meas_spar_vbe setup, and will only work if that measured S-parameter data exists. This setup and the meas_spar_vbe setup have identical sets of bias input variables. It is important that they remain the same: do not change the inputs.

1 Select the **extr_be_cap** setup. Display the **cbe_vs_vbe** plot.

2 Select **Extract/Optimize**.

3 Select **sdta > Execute**. This reads the measured data, de-embeds the extrinsics, and transforms the intrinsic data into a table of bias-dependent small-signal element values. This is the data on which the rest of the transforms are performed.

4 In the Parameters table, set EEBJT.tf, xtf, and itf to 0.000.

5 Scroll the Parameters table to show EEBJT2.cje, vje, and mje.

6 Select **cbe_mod > Execute**. This computes the base-emitter capacitance for the EEBJT2 model.

7 Select **set_cje > Execute**. This estimates the zero-bias junction capacitance parameter EEBJT2.cje from a direct extraction.

8 Select **cbe_mod > Execute** again.

9 Select **opt_mje_vje > Execute**. This optimizes mje and vje only, holding cje constant.

10 Select **set_opt_vars > Execute**. This sets the optimization limit variables for cje and vje.

11 Select **opt_cje_mje_vje > Execute**. This optimizes the three junction capacitance parameters by minimizing the error between the measured Cbe and the modeled Cbe. You can see the values change in the Parameters table.

Extracting Parameters with extr_bc_cap (AC)

This setup extracts the base-collector junction depletion capacitance parameters EEBJT2.cjc, vjc, and mjc. It uses the measured S-parameters from the meas_spar_vbc setup, and will only work if that measured S-parameter data exists. Note that this setup and the meas_spar_vbc setup have identical sets of bias input variables. It is important that they remain the same: do not change the inputs.

1 Select the **extr_bc_cap** setup. Display the **cbc_vs_vbc** plot.

2 Scroll the Parameters table to show EEBJT2.cjc, vjc, and mjc.

3 Select **Extract/Optimize**.

4 Select **sdta > Execute**. This reads the measured data, de-embeds the extrinsics, and transforms the intrinsic data into a table of bias-dependent small-signal element values. This is the data on which the rest of the transforms are performed.

5 Select **cbc_mod > Execute**. This computes the base-collector capacitance for the EEBJT2 model.

6 Select **set_cjc > Execute**. This estimates the zero-bias junction capacitance parameter EEBJT2.cjc from a direct extraction.

7 Select **opt_mjc_vjc > Execute**. This optimizes mjc and vjc only, holding cjc constant.

8 Select **set_opt_vars > Execute**. This sets the optimization limit variables for cjc and vjc.

9 Select **opt_cjc_mjc_vjc > Execute**. This optimizes the three junction capacitance parameters by minimizing the error between the measured Cbc and the modeled Cbc. You can see the values change in the Parameters table. The following figure shows an example of the plotted data following optimization.
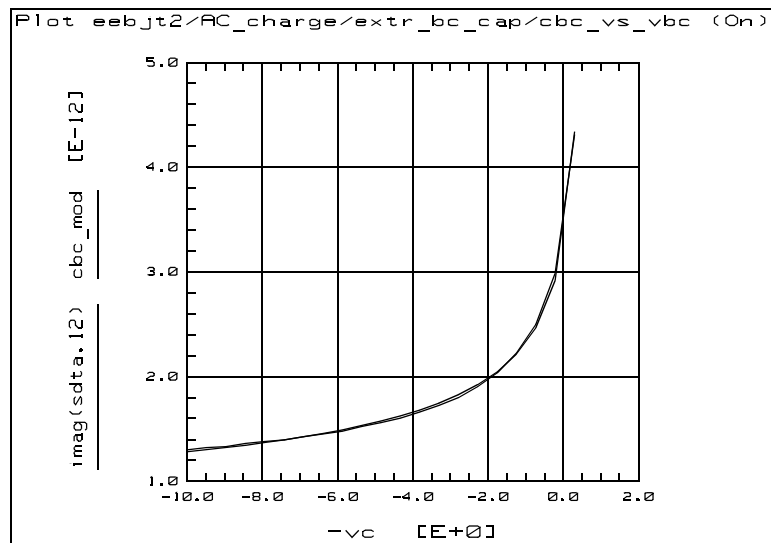
Figure 160 Example cbc_vs_vbc Plot

Extracting Parameters with extr_be_delay (AC)

This setup extracts the base-emitter junction transit time parameters EEBJT2.tf and itf. It uses the measured S-parameters from the meas_spar_vbe_fb setup, and will only work if that measured S-parameter data exists. Note that this

setup and the meas_spar_vbe_fb setup have identical sets of bias input variables. It is important that they remain the same: do not change the inputs.

1 Select the **extr_be_delay** setup. Display the **ft_vs_ic** plot. The following figure shows an example of ft plotted with respect to collector current in the ft_vs_ic plot.
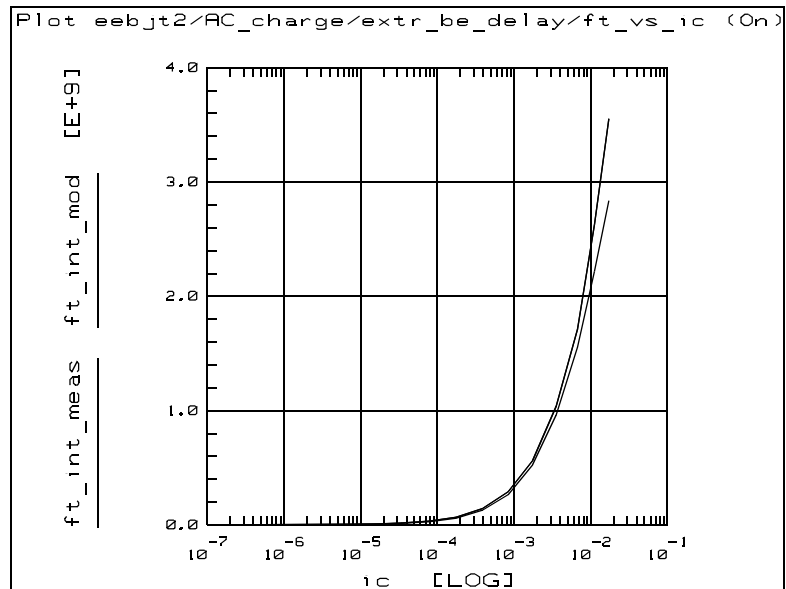
Figure 161 Example ft_vs_ic Plot

2 Scroll the Parameters table to display EEBJT2.tf and itf. (They are not together in the table.)

3 Select **Extract/Optimize**.

4 Select **sdta > Execute**. This reads the measured data, de-embeds the extrinsics, and transforms the intrinsic data into a table of bias-dependent small-signal element values. This is the data on which the rest of the transforms are performed.

5 Select **cbe_mod > Execute**. This computes the base-emitter capacitance for the EEBJT2 model.

6 Select **opt_cbe > Execute**. This optimizes the parameters
   EEBJT2.tf and itf by minimizing the error between the
   measured Cbe and the modeled Cbe. The values in the
   Parameters table will probably change by only a small
   amount.

7 Select **ft_int_meas > Execute**. This computes the intrinsic
   device Ft from the measured gm and Cbe.

8 Select **ft_int_mod > Execute**. This computes the intrinsic device
   Ft from the modeled gm and Cbe.

Extracting Parameters with extr_vtf (AC)

This setup extracts the base-emitter junction transit time
parameter EEBJT2.vtf. It uses the measured S-parameters from
the meas_spar_vce_ib setup, and will only work if that
measured S-parameter data exists. Note that this extr_vtf setup
and the meas_spar_vce_ib setup have identical sets of bias
input variables. It is important that they remain the same: do
not change the inputs.

1 Select the **extr_vtf** setup. Display the **cbe_vce** plot.

2 Scroll the Parameters table to show EEBJT2.vtf (near the
   bottom of the table).

3 Select **Extract/Optimize**.

4 Select **sdta > Execute**. This reads the measured data,
   de-embeds the extrinsics, and transforms the intrinsic data
   into a table of bias-dependent small-signal element values.
   This is the data on which the rest of the transforms are
   performed.

5 Select **cbe > Execute**. This computes the base-emitter
   capacitance for the EEBJT2 model.

6 Select **opt_vtf > Execute**. This optimizes the parameter
   EEBJT2.vtf by minimizing the error between the measured
   Cbe and the modeled Cbe. The value in the Parameters table
   will probably change by only a small amount.

Validating the Model with the AC_validation DUT

The AC_validation DUT contains the setup spars_fwd_active to validate the fit of the nonlinear model to S-parameters measured over bias. No parameters are extracted in this setup.

The bias inputs are preset using existing variables that represent typical bias values. You can alter these bias sweeps in order to examine the model fit at any bias. However, you should not change the values of existing variables. Instead, use new variables or enter numerical values directly into the inputs.

None of the macros is equivalent in any way to this setup.

1 Select the **spars_fwd_active** setup. Open the plots and arrange them on the screen so that you still have access to the Model window.

2 Select **Extract/Optimize**.

3 Select **y_int_mod > Execute**. This computes the intrinsic modeled Y-parameters of the EEBJT2 model over the bias range specified by the inputs. It also automatically calls the s_mod transform, which converts the intrinsic Y-parameters to S-parameters and adds the effects of the extrinsic elements to the S-parameters.

4 Observe the modeled data converge to the measured data in the plots.

5 If you want to save the parameters in the Parameters table in an MDIF (Measurement Data Interchange Format) file for compatibility with Libra, **Execute** the **write_pars_to_file** transform and type your own filename for the parameters. A copy of the MDIF file is written to the IC-CAP Status window.

References

1   *Modeling the Bipolar Transistor.* Ian Getreu. 1976 Tektronix Inc.

Direct Extraction of GaAs MESFET Intrinsic Element and Parasitic Inductance Values. Eric Arnold, Michael Golio, Monte Miller, and Bill Beckwith. IEEE MTT-S Microwave Symposium Digest (Dallas) 1990.

8
Mextram 504 Model

A new version of the Philips model for vertical bipolar transistor has been developed—Mextram level 504. This model provides a better description than Mextram level 503 for first and high-order derivatives of the characteristics. Special attention was paid to SiGe transistors, and self-heating was included in the description. A complete description of the Mextram 504 model can be found in the following Philips report:

> *The Mextram Bipolar Transistor Model*
> level 504
> J.C.J. Paasschens, W.J. Kloosterman

This report is available as a pdf file and can be downloaded from the following web site:

http://www.semiconductors.philips.com/Philips_Models/
bipolar/mextram/

Agilent Technologies

Mextram 504 in IC-CAP

The IC-CAP example file for the Mextram 504 model extraction is located at:

/examples/model_files/bjt/mxt504_npn.mdl

The example file refers to a four-terminal device. The extraction procedure and the dedicated IC-CAP transforms used in the example file are extensively documented in the following report from Philips:

Parameter Extraction for the Bipolar Transistor Model Mextram
level 504
J.C.J. Paasschens, W.J. Kloosterman, R.J. Havens

This report is available as a pdf file and can be downloaded from the following web site:

http://www.semiconductors.philips.com/Philips_Models/ bipolar/mextram/

A list of the IC-CAP transforms used in the Mextram 504 extraction can also be found in Chapter 8, "IC-CAP Functions," in the *IC-CAP Reference* manual.

Additional References for the Mextram Transistor Model

- Latest information about Philips Models:

  http://www.semiconductors.philips.com/Philips_Models/
  news_flash/

- News about Mextram 504:

  http://www.semiconductors.philips.com/Philips_Models/
  news_flash/archive/#reports/

- Additional reports on Mextram (includes a list of published
  papers and references)

  http://www.semiconductors.philips.com/Philips_Models/
  bipolar/mextram/

9
VBIC Model

The VBIC vertical BJT model has been developed specifically as a replacement for the SPICE Gummel-Poon model, by representatives of the IC and CAD industries. The SPICE Gummel-Poon model has remained essentially an unchanged industry standard for over 20 years. For devices available until recent times it has been an intuitive and physically consistent formulation that has generally met the requirements of BJT device modeling.

However, improvements in modern device technology have increasingly indicated a need for improvements in BJT modeling techniques. Some of the technology changes are these:

- Reduced base width.
- Smaller device dimensions and changes in device structure.
- Changes in doping profiles.
- Parasitic substrate transistor.

Reduced base width means that higher-order effects have more significance. The Early effect approximation in the Gummel-Poon model is not sufficiently accurate to account for these effects.

Agilent Technologies

Smaller device dimensions increase the importance of parasitic overlap capacitances not accounted for by the Gummel-Poon model. As the base width becomes smaller, the response becomes more nonlinear.

Changes in doping profiles increase the quasi-saturation effects of collector resistance, which is not accounted for in the Gummel Poon model.

The parasitic substrate transistor is not included in the Gummel-Poon model. Therefore both DC and AC modeling are less accurate than needed in today's devices.

To overcome these issues, representatives from the IC and CAD industries have collaborated over the last few years to recommend an improved standard BJT model for the semiconductor industry. The result is VBIC, a model that is now viable and will continue to develop as further improvements are made.

VBIC (vertical bipolar inter-company model) includes improved modeling of the Early effect (output conductance), substrate current, quasi-saturation, and behavior over temperature: information necessary for accurate modeling of current state-of-the-art devices. However, it has additionally been defined so that, with default parameters, the model will simplify to be as similar as possible to the Gummel-Poon model.

Advantages of VBIC over the Gummel-Poon model include the following:

- An Early effect model based on the junction depletion charges.

- A modified Kull model for quasi-saturation valid into the Kirk regime (the high-injection effect at the collector).

- Inclusion of the parasitic substrate transistor.

- An improved single-piece junction capacitance model for all three junction capacitances.

- Improved static temperature scaling.

- First-order modeling of distributed base and emitter AC and DC crowding.

- Overall improved high-level diffusion capacitance modeling (including the quasi-saturation charge).

- Inclusion of parasitic overlap capacitances.

- Inclusion of the onset of weak avalanche current for the base-collector junction.

- High-order continuity (infinite) in equations.

- A noise model similar to that of the Gummel-Poon model, with shot, thermal, and 1/f components.

- A defined self-heating model with hooks included in code: the self-heating model itself will be available in a future release of VBIC.

The Agilent 85193D VBIC model assumes the device under test to be a vertical bipolar NPN transistor, fabricated in a P-type substrate.

NOTE	The example VBIC model is set up to test an NPN transistor. With certain modifications, you can use the example model to extract a PNP transistor. Set the model variable *polarity* to PNP, and adjust the bias values in the Setups to appropriately bias a PNP device.

Getting Started in IC-CAP

The procedures in this chapter assume the following:

- The system has been set up and switched on according to the instructions under:
  - Agilent 8510-based system—"Configuring the System Hardware" on page 575 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - Agilent 8753-based system—"Configuring the System Hardware" on page 609 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"
  - Wiltron 360-based system—Refer to your Wiltron documentation

- The calibration standards have been removed from their containers to allow them to reach ambient room temperature.

- The IC-CAP software has been properly installed on the computer.

- IC-CAP has been configured to recognize the system hardware, and the SMUs have been renamed (as described in "Renaming the SMUs" on page 465), based on the instructions in the appropriate chapter:
  - "Performing Hardware Setup in IC-CAP" on page 585 in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"
  - "Performing Hardware Setup in IC-CAP" on page 617 in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Refer to Appendix A, "Agilent 85190A IC-CAP" for instructions on launching the IC-CAP software and opening the model file you need. The VBIC model file is under:

/examples/model_files/bjt/vbic_npn.mdl

Renaming the SMUs

Use this procedure to rename the SMUs to match the names used in the measurement setups, so that the software can control the bias supplies.

When you run IC-CAP, the software initially identifies the Agilent 4142B DC source/monitor SMUs according to the numbers of the slots in which they are installed. A medium-power SMU occupies one slot. A high-power SMU occupies two slots, and is identified by the higher slot number of the two.

A measurement system can include different combinations of SMUs. This model requires the use of four SMUs.

In configuring IC-CAP to recognize the measurement system, set the SMU names in IC-CAP to identify the SMU connections at the measurement terminals. In the VBIC model, the SMUs are identified as follows:

VB	Base	VE	Emitter
VC	Collector	VS	Substrate

To change the SMU names in IC-CAP, select **Configure** under Instrument List in the hardware window. A window is displayed similar to the one in the illustration.
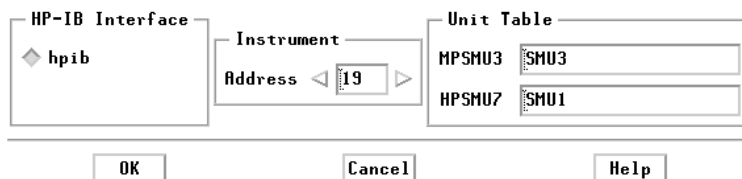
Figure 162   Example Configuration of SMUs in the Agilent 4142

In the Unit Table, change the existing SMU names in the box to the names required by VBIC. (If you use the GNDU SMU for the emitter connection, change its name to VE.) Select **OK** when you are done.

If you wish to save the hardware configuration, select **File > Save As**, and assign a filename with the suffix *.hdw*.

Using the VBIC Model

This chapter provides a sequence of example procedures that take you step-by-step through a VBIC measurement and extraction. To facilitate the model extraction, the parameters to be extracted are divided into subsets, where each subset is extractable from a single set of measured data using explicit bias conditions. This speeds the total time of parameter extraction.

A series of different IC-CAP DUTs and setups are used to measure current, voltage, capacitance, or S-parameters under different bias and frequency conditions.

The procedures are documented in the proper sequence to allow earlier measured data or extracted parameters to be used as a basis for later measurements or extractions.

NOTE	These procedures measure and model a vertical NPN device.

The procedures in this chapter include instructions for the following:

- Getting started in IC-CAP 5.0 and opening the model.
- Configuring IC-CAP to recognize the measurement system.
- Setting hardware instrument states (instrument options).
- Setting the inputs for each measurement.
- Performing the capacitance, DC, and S-parameter measurements.
- Storing the measured data.
- Initializing device parameters using a macro.
- Performing a total model extraction using a single macro.
- Performing individual extractions using lesser macros.
- Also provides information on the transforms in the individual setups.

• Storing the model.

The procedures start with example values for the measurement inputs. They provide guidelines on setting the values to measure your device. However, the values you use will be based on your knowledge of your device and on information provided in its data sheet, as well as on the guidelines provided here: they may or may not correspond with the example. In some cases, it may be necessary to set the inputs, make a measurement, and then reset the inputs based on the measurement results. The procedures give suggestions on judging the applicability of the input values and measured data.

The illustrations of plotted data are provided as visual examples of possible results. They are not intended to represent a sequential measurement of a single device, nor do they necessarily correspond with the example measurement inputs described in the procedures. The plots can be used to check the reasonableness of your own data, which should appear similar in shape but not necessarily coincident in values.

In addition, the IC-CAP software itself includes measured data for some or all of the setups. This data shows additional examples of possible results. Many, though not necessarily all, of the input settings for this measured data correspond with the example settings in this manual.

Model Parameters

The table in the next few pages lists the VBIC model parameters and their definitions.

Table 51   VBIC Model Parameters

Parameter	Definition	Default Value
TNOM	Nominal measurement temperature of parameters	2.7E+01
RCX@TNOM	Extrinsic collector resistance	0.0
RCI@TNOM	Intrinsic collector resistance	0.0
VO@TNOM	Epi drift saturation voltage	0.0

Table 51   VBIC Model Parameters (continued)

Parameter	Definition	Default Value
GAMM@TNOM	Epi doping parameter	0.0
HRCF	High-current RC factor	1.0
RBX@TNOM	Extrinsic base resistance	0.0
RBI@TNOM	Intrinsic base resistance	0.0
RE@TNOM	Emitter resistance	0.0
RS@TNOM	Substrate resistance	0.0
RBP@TNOM	Parasitic base resistance	0.0
IS@TNOM	Transport saturation current	1.0E−16
NF@TNOM	Forward emission coefficient	1.0
NR@TNOM	Reverse emission coefficient	1.0
FC	Forward bias depletion capacitance limit	0.9
CBEO	Extrinsic base-emitter overlap capacitance	0.0
CJE@TNOM	Base-emitter zero-bias capacitance	0.0
PE@TNOM	Base-emitter built-in potential	0.75
ME	Base-emitter grading coefficient	0.33
AJE	Base-emitter capacitance smoothing factor	−0.5
CBCO	Extrinsic base-collector overlap capacitance	0.0
CJC@TNOM	Base-collector intrinsic zero-bias capacitance	0.0
QCO	Epi charge parameter	0.0
CJEP@TNOM	Base-collector extrinsic zero-bias capacitance	0.0
PC@TNOM	Base-collector built-in potential	0.75
MC	Base-collector grading coefficient	0.33
AJC	Base-collector capacitance smoothing factor	−0.5
CJCP@TNOM	Substrate-collector zero-bias capacitance	0.0

Table 51 VBIC Model Parameters (continued)

Parameter	Definition	Default Value
PS@TNOM	Substrate-collector built-in potential	0.75
MS	Substrate-collector grading coefficient	0.33
AJS	Substrate-collector capacitance smoothing factor	−0.5
IBEI@TNOM	Ideal base-emitter saturation current	1.0E−18
WBE	Portion of IBEI from Vbei, 1−WBE from Vbex	1.0
NEI	Ideal base-emitter emission coefficient	1.0
IBEN@TNOM	Non-ideal base-emitter saturation current	0.0
NEN	Non-ideal base-emitter emission coefficient	2.0
IBCI@TNOM	Ideal base-collector saturation current	1.0E−16
NCI	Ideal base-collector emission coefficient	1.0
IBCN@TNOM	Non-ideal base-collector saturation current	0.0
NCN	Non-ideal base-collector emission coefficient	2.0
AVC1	Base-collector weak avalanche parameter 1	0.0
AVC2@TNOM	Base-collector weak avalanche parameter 2	0.0
ISP@TNOM	Parasitic transport saturation current	0.0
WSP	Portion of ICCP from Vbep, 1−WSP from Vbci	1.0
NFP	Parasitic forward emission coefficient	1.0
IBEIP@TNOM	Ideal parasitic base-emitter saturation current	0.0
IBENP@TNOM	Non-ideal parasitic base-emitter saturation current	0.0
IBCIP@TNOM	Ideal parasitic base-collector saturation current	0.0
NCIP	Ideal parasitic base-collector emission coefficient	1.0

Table 51   VBIC Model Parameters (continued)

Parameter	Definition	Default Value
IBCNP@TNOM	Non-ideal parasitic base-collector saturation current	0.0
NCNP	Non-ideal parasitic base-collector emission coefficient	2.0
VEF	Forward Early voltage. Zero represents infinity.	0.0
VER	Reverse Early voltage. Zero represents infinity.	0.0
IKF	Forward knee current. Zero represents infinity.	0.0
IKR	Reverse knee current. Zero represents infinity.	0.0
IKP	Parasitic knee current. Zero represents infinity.	0.0
TF	Forward transit time	0.0
QTF	Variation of TF with base-width modulation	0.0
XTF	Coefficient of TF bias dependence	0.0
VTF	Coefficient of TF dependence on Vbc	0.0
ITF	Coefficient of TF dependence on Ic	0.0
TR	Reverse transit time	0.0
TD	Forward excess-phase delay time	0.0
KFN	Base-emitter flicker noise constant	0.0
AFN	Base-emitter flicker noise exponent	1.0
BFN	Base-emitter flicker noise 1/f dependence	1.0
XRE	Temperature exponent of emitter resistance	0.0
XRB	Temperature exponent of base resistance	0.0
XRC	Temperature exponent of collector resistance	0.0

Table 51 VBIC Model Parameters (continued)

Parameter	Definition	Default Value
XRS	Temperature exponent of substrate resistance	0.0
XVO	Temperature exponent of VO	0.0
EA	Activation energy for IS	1.12
EAIE	Activation energy for IBEI	1.12
EAIC	Activation energy for IBCI/IBEIP	1.12
EAIS	Activation energy for IBCIP	1.12
EANE	Activation energy for IBEN	1.12
EANC	Activation energy for IBCN/IBENP	1.12
EANS	Activation energy for IBCNP	1.12
XIS	Temperature exponent of IS	3.0
XII	Temperature exponent of IBEI / IBCI / IBEIP /IBCIP	3.0
XIN	Temperature exponent of IBEN / IBCN /IBENP /IBCNP	3.0
TNF	Temperature coefficient of NF	0.0
TAVC	Temperature coefficient of AVC2	0.0
RTH	Thermal resistance	0.0
CTH	Thermal capacitance	0.0

The DUTS and Setups

The DUTs and setups are the central focus for the measurement and extraction procedures. The DUTs (for example cap), indicated by the inverted triangles, are groupings of similar measurement setups used to extract related model parameters. The setups (for example Cbe) contain the information used to define the inputs and outputs for each particular measurement. The following figure shows a portion of the DUT/Setup panel.
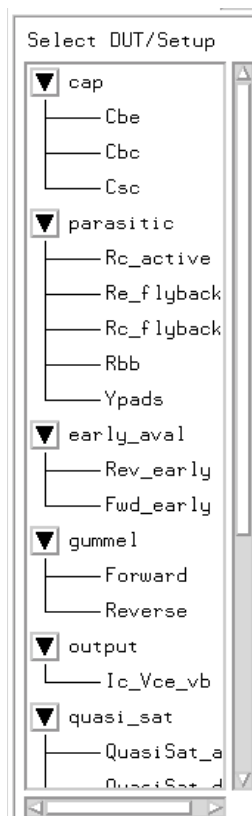
Figure 163   The DUT/Setup Panel

The following table summarizes the parameters that will be extracted and the IC-CAP DUTs and setups with which they are associated.

Table 52   Summary of Setups and Parameters Extracted

DUT	Setup	Parameters
cap	Cbe	CJE, PE, ME
	Cbc	CJC, PC, MC

Table 52   Summary of Setups and Parameters Extracted (continued)

DUT	Setup	Parameters
	Csc	CJCP, PS, MS
parasitic	Rc_active	RCX
	Re_flyback	RE
	Rc_flyback	RCX
	Rbb	RBX, RBI, WBE
	Ypads	none
early_aval	Rev_early	CJE, WBE, QBO
	Fwd_early	VEF, VER, AVC1, AVC2, CJE, CJEP
gummel	Forward	IS, NF, IBEI, NEI, IBEN, NEN
	Reverse	NR, IBCI, NCI, IBCN, NCN, ISP, NFP, IKR
output	Ic_Vce_vb	HRCF, VO, QCO, GAMM, IKF, RCI, RCX
quasi_sat	QuasiSat_ac	HRCF, VO, QCO, GAMM, IKF, RCI, RCX
	QuasiSat_dc	HRCF, VO, QCO, GAMM, IKF, RCI, RCX
	Ypads	none
sparm_cap	Cbe	CJE, PE, ME
	Cbc_Csc	CJC, PC, MC, CJCP, PS, MS
	Cpads	none
delay	Ftvsic	TF, XTF, ITF, VTF, QTF
	Forward_Tau	TF, XTF, ITF, VTF, QTF
	Reverse_Tau	TR, QCO, IKP
	Ypads	none

Measurement Procedures

All the measurements are made before any extractions are performed. It is then possible to extract the parameters automatically from the measured data using the macro procedures. This will be explained later in the chapter, following the measurement procedures.

Defining the Instrument States

For each of the measurement setups, it is necessary to define the instrument states (options) for the measurement instruments that will be used in that setup. The instrument states need to be defined individually for each setup. Different setups use different measurement instruments. However, with minor exceptions that will be explained, the instrument states for the same measurement type must be used for all the setups that use that measurement type.

1 After you have selected a setup, for example gummel Forward, the setup window tabs are displayed.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states for all instruments used in that measurement. For example, in the DC measurement setups, the instrument states for the DC source/monitor will be displayed. In the CV meter capacitance measurement setups, the instrument states for the CV meter or LCR meter in use will be displayed. In the S-parameter measurement setups, the instrument states for both the DC source/monitor and the network analyzer will be displayed.

The instrument state settings will be explained in this chapter with their corresponding setups.

Making the cap Cbe Measurement

The cap setups use a CV meter to measure the junction capacitances. This setup measures the base-emitter junction capacitance.

NOTE

You need only do *one* set of capacitance measurements, using either the cap setups and a CV meter, or the sparm_cap setups and a network analyzer. The network analyzer measurements provide greater accuracy. Also you may find it more convenient to use the network analyzer, since it must be calibrated for other measurements anyway.

Parameters

The data from this measurement will be used to extract these parameters:

CJE	Base-emitter zero-bias capacitance
PE	Base-emitter built-in potential
ME	Base-emitter grading coefficient

Defining the CV Meter Instrument State

1 Select the **Instrument Options** tab, and a window is displayed showing the instrument states for the measurement instrument used in this setup.

2 Several different CV meters and LCR meters can be used with IC-CAP. Refer to Chapter 1, "Supported Instruments," of the *IC-CAP Reference* manual for specific guidelines on setting the instrument states for your CV or LCR meter.

3 Note that IC-CAP provides instrument drivers only for those CV meters and LCR meters that are listed in the "Measurement" chapter.

The Inputs and Outputs

1 In the cap DUT select the **Cbe** setup. The setup inputs and outputs are displayed.

   • The vbe base-emitter voltage values are set to sweep from approximately VBEmax to 0.4V.

   • The output is cbe, the base-emitter capacitance.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Cbe_Vbe plot is displayed.

2 Select the **Measure/Simulate** tab, then click **Measure**.

3 You can save the data using **File > Save As** and a filename with the suffix *.set*. Then click **OK**.

Making the cap Cbc Measurement

This setup measures the base-collector junction capacitance, using a CV meter.

Parameters

The data from this measurement will be used to extract these parameters:

CJC	Base-collector intrinsic zero-bias capacitance.
PC	Base-collector built-in potential.
MC	Base-collector grading coefficient.

Defining the CV Meter Instrument State

Follow the same process you used in the cap Cbe measurement. Use the same settings.

The Inputs and Outputs

From the cap DUT, select the Cbc setup. The setup inputs and outputs are displayed.

- The *vbc* base-collector voltage values are set to sweep from approximately VBCmax to 0.4V.
- The output is cbc, the base-collector capacitance.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Cbc_Vbc plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the cap Csc Measurement

This setup measures the substrate-collector junction capacitance, using a CV meter.

Parameters

The data from this measurement will be used to extract these parameters:

CJCP   Substrate-collector zero-bias capacitance

PS   Substrate-collector built-in potential

MS   Substrate-collector grading coefficient

Defining the CV Meter Instrument State

Follow the same process you used in the cap Cbe measurement. Use the same settings.

The Inputs and Outputs

From the cap DUT, select the Csc setup.

- The vsc substrate-collector voltage values are set to sweep from approximately VBCmax to 0.4V.
- The output is csc, the substrate-collector capacitance.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Csc_Vsc plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the parasitic Rc_active Measurement

This uses the fact that the substrate current increases with an increasing voltage drop across the constant portion of the collector resistor when the device is in either quasi-saturation or saturation. Because the noise level of the substrate current is often quite high, the dynamic range of this method may not always be sufficient to resolve the bias-independent collector resistance adequately.

This method is described in detail in Jorg Berkner's *A Survey of DC Methods for Determining the Series Resistance of Bipolar Transistors Including the New Delta ISub Method* (see Bibliography).

Parameters

The data from this measurement will be used to extract this parameter:

RCX   Extrinsic collector resistance

Defining the DC Source/Monitor Instrument State

This procedure configures IC-CAP with the correct instrument state settings (instrument options) to control the DC source/monitor.

You will need to set the DC source/monitor instrument state separately for *each* measurement setup that uses it. However, use the same settings for all the measurement setups (with the exception of Integ Time, which is explained below).

The settings used here are examples. Your settings may differ. Explanations for the different settings are listed below.

1 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states for the DC source/monitor.

2 Set Use User Sweep to No, as it is unnecessary for these procedures, and the source/monitor internal sweep is faster.

3 Hold Time is the delay in seconds before starting a sweep, to allow for DC settling. Generally, no hold time is required.

4 Delay Time is the amount of time in seconds the instrument waits before measuring at each step of an internal or user sweep. It can generally be set to 0.0.

5 For Integ Time, M (medium) is a good default choice. You can speed up a measurement by using S (short), but this is not recommended because it degrades the dynamic range of the measurement. In measurements where a long integration time is needed for noise reduction, use L (long).

6 Set Range to 0 to implement SMU auto-ranging.

7 Power Compliance is used to set the maximum current/voltage combination for the DC source/monitor. However, in the model, SMU voltage and current compliances are set individually in the individual setups. Therefore the value here can be set to 0.000.

8 Set SMU Filters ON to Yes. This switches in low-pass filters on the SMU outputs, to protect the device from voltage spikes caused by DAC output changes.

Figure 164 Instrument States for the Agilent 4142B DC Source/Monitor

9 The VBIC model does not use pulsed measurements, therefore Pulse Unit can be left blank. All other Pulse settings are then irrelevant and can be ignored.

10 Module Control is not used in these procedures. Leave the field blank.

11 Init Command sets the instrument to a mode not supported by other fields in this table. It is not used in this model. Leave the field blank.

12 If you wish, refer to the DC source/monitor manual for more detail.

The Inputs and Outputs

From the parasitic DUT, select the **Rc_active** setup.

- The ve emitter voltage is set to sweep from approximately −0.4V to −1.2V. The precise voltage values may vary. The intent is to measure low enough that the non-ideal base current effects are visible, and high enough in collector current to observe the compression due to high-level injection and/or quasi-saturation.

- The vc collector voltage is set to approximately −0.3V, so that the parasitic PNP is slightly forward biased. This is often necessary to pull the substrate current Is out of the noise floor associated with the substrate. If the collector resistance is too small, its associated voltage drop may be too small to adequately bias the parasitic PNP. The result is that the substrate current never becomes large enough to rise above the substrate noise floor.

- The vs substrate voltage is usually set at −1V to approximately −3V.

- The vb base voltage is set to ground (0.0V).

- The outputs are is, substrate current; ic, collector current; and ib, base current; measured with the DC source/monitor.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Is_Vbe plot is displayed.

2 Select **Measure/Simulate > Measure**. The measured data should resemble the illustration. The unusual vertical line on the trace is caused by the extraction algorithm, and will be explained in the extraction section later in this chapter.
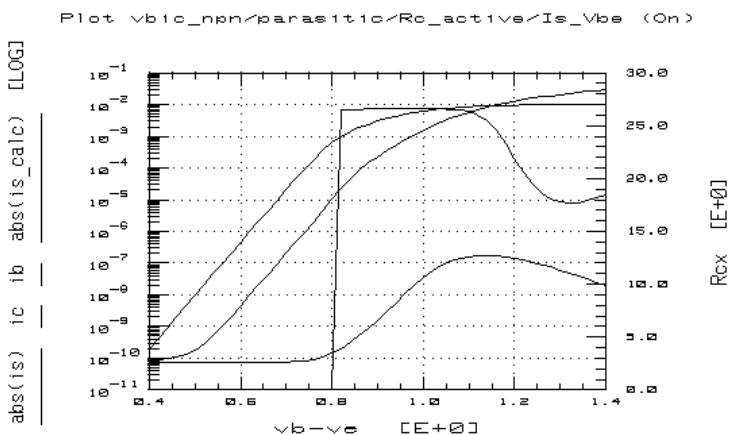
Figure 165   Example Plotted parasitic Rc_active Is_Vbe Data

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the parasitic Re_flyback Measurement

In this procedure a current source drives the base to negate the effect of the base parasitic resistance. The collector is open-circuited by setting its SMU current to zero, and the emitter is grounded. The change in collector voltage is monitored vs the change in base current. The collector voltage is the base current times the parasitic resistance in the emitter.

Parameters

The data from this measurement will be used to extract this parameter:

RE   Emitter resistance

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

Select the **Re_flyback** setup.

- The ib base current is set to sweep from approximately 10 mA to 50 mA (2 mA/um^2).
- The ve emitter voltage is set to ground (0.0V).
- The is substrate current is set to 0A, so that only the forced base current goes through the emitter.
- The ic collector current is set to 0A, so that the voltage at the collector can be measured without any voltage drop across the series collector resistance.
- The outputs are vc, collector voltage; ie, emitter current; and vb, base voltage.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Vc_Ie plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Making the parasitic Rc_flyback Measurement

This procedure is similar to the Re_flyback measurement. A current source drives the base to negate the effect of the base parasitic resistance. In this case the emitter is open-circuited by setting its SMU current to zero, and the collector is grounded. The change in emitter voltage is monitored vs the change in negative collector current. The emitter voltage is the base current times the parasitic resistance in the collector.

Parameters

The data from this measurement will be used to extract this parameter:

RCX   Extrinsic collector resistance

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

Select the **Rc_flyback** setup.

- The ib base current is set to sweep from approximately 10 mA to 50 mA (2 mA/um^2), the same as in the Re_flyback measurement.
- The vc collector voltage is set to ground (0.0V).
- The is substrate current is set to 0A, so that only the forced base current goes through the collector.
- The ie emitter current is set to 0A, so that the voltage at the emitter can be measured without any voltage drop across the series emitter resistance.
- The outputs are ve, emitter voltage; and ic, collector current.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Ve_Ic plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Calibrating the Network Analyzer

NOTE   This information is presented here because the next measurements, parasitic Rbb and Ypads, use S-parameter as well as DC measurements.

It is important to calibrate the network analyzer before you perform any S-parameter measurements. Good calibration of the network analyzer is critical to a good measurement and extraction. Without a good calibration, the results of the extraction *will* be inaccurate.

The VBIC measurement procedures require a swept broadband cal and at least one CW cal. (One is probably sufficient, but you may choose to do certain measurements at different frequencies. The example provided here uses one broadband cal and one CW cal frequency.) The calibrations can be done with one set of standards measurements, by making the CW frequency cal a subset of the broadband cal. More than one subset CW cal can be done if necessary.

The broadband cal is used for the parasitic Rbb and Ypads measurements, described next. CW cals are used for the QuasiSat_ac and quasi_sat Ypads measurements, and the delay measurements. Also, a CW cal is used for the sparm_cap measurements if you choose to do the junction capacitance measurements with a network analyzer rather than a CV meter. It is preferable to use a network analyzer, because it provides more accurate capacitance data at higher frequencies, and you already need to calibrate the network analyzer for other measurements. If you decide to use a CV meter, you will need to set it up and calibrate it in addition to the other hardware procedures.

Procedures

For the step-by-step calibration procedures, refer to:

- "Calibrating the Network Analyzer" on page 589 (for an Agilent 8510-based system) in Appendix B, "Using IC-CAP with an Agilent 85122A Precision Modeling System"

- "Calibrating the Network Analyzer" on page 631 (for an Agilent 8753-based system) in Appendix C, "Using IC-CAP with an Agilent 85123A Device Modeling System"

Set the frequency range for the parasitic swept broadband calibration to cover the operating frequency range of the device under test. The start frequency should be as low as possible, around 50 to 100 MHz. Set a number of points that will make a reasonable number of measurements across the sweep: 11 is probably sufficient.

For the CW measurements, especially the sparm_cap measurements, use a cal frequency as low as possible, approximately 50 to 200 MHz. This provides lower parasitics and better dynamic range for the junction capacitance measurements (for more detailed information, see below).

The frequency of the CW calibration subset or subsets must be at a calibrated point in the broadband calibration range. Use a measurement point equal to the start frequency plus an integer multiple of the step size. For example:

50 MHz (start freq) + 10 · 500 MHz (step size) = 5.05 GHz

Because of the large gain of a BJT device at low frequencies, it is important to make sure the input power from the network analyzer is not so large as to saturate the device. In setting the attenuation and power level, take care that the power will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. Attenuation and power level must be the same for the subset cals as for the broadband cal.

Note the numbers of the cal sets or registers where you save the calibrations: you will need to list one of these numbers as part of the instrument state for each of the S-parameter measurement setups.

After you perform the aforementioned cal procedures, return to this chapter and continue.

More on Power Levels and Capacitance Measurements

The primary factor to consider in understanding the available range of power is the S-parameter test set, and whether it is coupler-based or power-splitter-based. The test set generally used in a 26.5 GHz system is the Agilent 8515B, which uses a power splitter to couple signals to the detector. The "economy" version test set is the Agilent 8514B (20 GHz), which uses a directional coupler.

The directional coupler in the Agilent 8514B has a −20 dB/decade power rolloff below 500 MHz, with some signal loss above that. The result is that in the frequency range 45 MHz to 2 GHz the minimum detectable signal of the Agilent 8514B is

only −66 dBm. However, over the same frequency range, the minimum detectable signal of the Agilent 8515B is −101 dBm. (Above 2 GHz, both test sets can detect at least −95 dBm signals.)

The model of synthesizer used does not affect these differences. Any potential improvement due to the synthesizer is obscured by the effects of the test set.

Capacitance Junction Measurements

In measuring capacitance junctions, the signal level at the device is generally kept below −30 dBm to keep nonlinearity-induced errors below 5 percent. Capacitance measurements are typically made in the 50 to 100 fF region.

Why is it necessary to keep the signal level below −30 dBm for a capacitance measurement? In fact, it is *not* necessary. The device is biased so as not to have any gain, therefore the input samplers and the test set are not saturated, nor is the output of the device. Additionally, the power level set on the synthesizer is not the same as the power level arriving at the device. Depending on the frequency range and the test set, there can be anywhere from 15 to 30 dB of loss between the source and the device.

One method is to solve for the minimum resolvable capacitance, given an input power and a dynamic range. This should be done for both S21 (Cbc) and S11 (Cbe), because their respective resolutions are different.

Device input power −10 dBm.

Noise Floor −90 dBm.

Dynamic range >80 dB. (80 dB = 1E−04 linear)

S21 Related to Cbc

$$|S21| = \frac{2 \cdot Z_0}{(2 \cdot Z_0 + 1/\omega C)} = 1 \cdot 10^{-04}$$

$$C = \frac{|S21|}{(2 \cdot \omega \cdot Z_0 (1 - |S21|))}$$

= *1.5917E-07/freq*

=> minimum C @ 100 MHz = 1.59 fF

S11 Related to Cbe

$$|S11| = \frac{(1/j\omega C) || (Z_0 - Z_0)}{(1/j\omega C) || (Z_0 + Z_0)} = 1 \cdot 10^{-04}$$

$$C = \frac{2|S11|}{Z_0 \cdot \omega(|S11| + 1)}$$

= *6.36556E-07/freq*

=> minimum C @ 100 MHz = 6.37 fF

This analysis verifies the observation that Cbc measurements are cleaner than Cbe measurements.

With an Agilent 8515B test set the same capacitance would put the signal 40 dB above the noise floor, which would allow 20 dB of leeway even as low as 100 MHz.

This verifies the initial statement that it is the S-parameter test set that determines the available range of power, and not the synthesized sweeper.

Instrument States for Swept S-Parameter Measurements

This procedure configures IC-CAP with the correct instrument state settings (instrument options) to control both the DC source/monitor and the network analyzer for a swept S-parameter measurement.

You need to set the instrument states separately for *each* measurement setup. However, the parasitic Rbb and Ypads setups are the only ones in this model that use a

swept-frequency S-parameter measurement. Other setups use CW frequency measurements, which are different and will be explained later.

The settings used here are examples. Your settings may differ. Explanations for the different settings are listed below.

Figure 166   Instrument States for Swept S-Parameter Measurements

NOTE	Do not use these settings for a CW S-parameter measurement. The settings for a CW measurement are different, and are documented later.

1   Select the **Instrument Options** tab, and a window will be displayed showing the instrument states.

2   Set the Agilent 4142 instrument states the same as for the DC measurements.

3   Set the Agilent 8510 instrument states according to the actual hardware configuration, the calibration used, and the device to be measured, following these guidelines.

4   Set Use User Sweep to No, as this is a standard sweep.

5   Hold Time is the time in seconds before the instrument starts a sweep, to allow for DC settling. Generally, no hold time is required.

6   Delay Time is the delay in seconds the instrument waits before setting each frequency in user sweep mode. You can set a delay, but it is generally not necessary.

7   Set Port 1 Atten and Port 2 Atten to the same levels in dB that you set in the network analyzer calibration. If an **IF OVERLOAD** error message is displayed during the measurement you will need to reduce the source output power or increase the attenuation, and calibrate again.

8   Set Source Power to the same level as in the network analyzer calibration.

9   If you calibrated with a power slope, you can set the corresponding value for Power Slope, to better view the response of a device with power dropoff at higher frequencies. The units for power slope are dB/GHz; the default value is 0.000. (A network analyzer message will caution you that the correction *may* be invalid, but this can be ignored.)

10   Set Fast Sweep (RAMP) to No, because the analyzer is in stepped-sweep mode, set previously in the calibration procedure.

11   Sweep Time applies only to ramp sweep mode, therefore the value set is irrelevant for a stepped-sweep measurement.

12 Set Use Fast CW to `Yes`, to minimize repeated switching between the test set ports.

13 Set Trim Sweep to `0`. This feature is used only in ramp sweep mode.

14 Set Avg Factor to the same averaging factor you set in the calibration. A good default averaging factor is 256. To speed your measurements, you may find it convenient to set an averaging factor as low as 16.

15 Set Cal Type (SHN) to `H`, for hardware.

16 Set Cal Set No to the cal set number in the analyzer where you stored your swept broadband frequency calibration. This example uses cal set 1.

17 Soft Cal Sequence refers to the sequence of measurements of the cal standard devices: load-open-short-thru.

18 Delay for Timeouts can generally be set to the default value of 0.000.

19 Set Use Linear List to `No`, because this is a standard stepped-frequency measurement.

20 Init Command sets the instrument to a mode not supported by other fields in this table. It is not used in this model. Leave the field blank.

21 If necessary, refer to the Agilent 8510C *Operating and Programming Manual* for more detail.

Making the parasitic Rbb Measurement

In this setup the base voltage is swept over a range such that the base-emitter junction varies from reverse biased to slightly forward biased. The collector voltage is set high enough to avoid saturation and quasi-saturation effects, and low enough to avoid breakdown effects. S-parameters are measured over a wide frequency range, in order to observe the distributed nature of the base resistance.

Parameters

The data from this measurement will be used to extract these parameters:

RBX	Extrinsic base resistance
RBI	Intrinsic base resistance
WBE	The portion of IBEI from Vbei, 1-WBE from Vbex

Defining the Instrument States

Follow the process described in "Instrument States for Swept S-Parameter Measurements" on page 489.

The Inputs and Outputs

Select the **Rbb** setup.

- The vb base voltage is swept over a range such that the base-emitter junction varies from reverse biased to slightly forward biased.
- The ve emitter voltage is set to ground (0.0V).
- The vs substrate voltage is set to ground (0.0V).
- The vc collector voltage is set to a value high enough to avoid saturation and quasi-saturation effects, and low enough to avoid breakdown effects.
- The freq frequency value is swept over a wide range, in order to observe the distributed nature of the base resistance.

| NOTE | If you set a MHz value, write *MEG* in full. If an error message is displayed, you may have entered *M* only. The outputs are S-parameters (*s*). |

Measuring and Plotting

1 From the Plots tab, select **Display All**. All the plots for this setup are displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the parasitic Ypads Measurement

This setup is used to measure an open dummy structure. With the device disconnected, the S-parameters are measured across the same swept frequency range as in the Rbb measurement. The data is used to de-embed the device S-parameters.

Parameters

No parameters are extracted from this data.

Defining the Instrument States

Follow the same process you used for the parasitic Rbb measurement, described in "Instrument States for Swept S-Parameter Measurements" on page 489. Use the same settings.

The Inputs and Outputs

From the parasitic DUT, select the **Ypads** setup.

• No voltages are defined or required.

• The frequency range defined in freq must be the same as you used in the Rbb setup, with the same number of points.

• The output is S-parameters (s), measured with the network analyzer.

Measuring and Plotting

1 Remove the test fixture from the measurement setup, or lift the probes of a probe station to present an open circuit.

2 From the Plots tab, select **Display Plot**. The Ypads/caps plot is displayed.

3 Select **Measure/Simulate > Measure**. The measured data should be constant with frequency, as in the illustration. If not, there may be a problem with the measurement, the calibration, or the test structure.
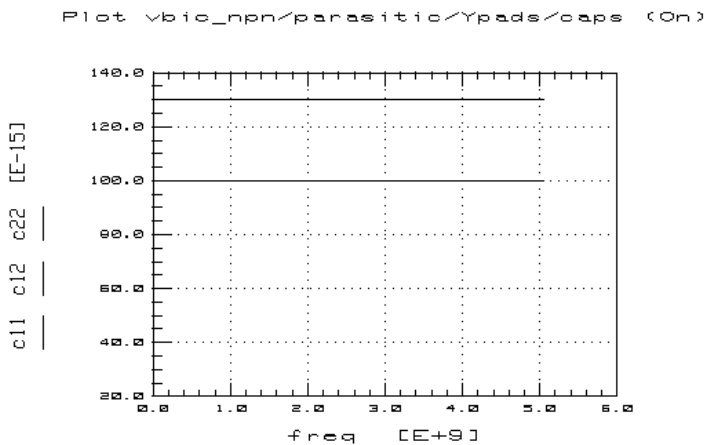
Figure 167   Example Plotted parasitic Ypads Data

4 Save the data using **File > Save As** and a filename with the suffix .*set*.

Making the early_aval Rev_early Measurement

The reverse early measurement models the effect of base-width modulation due to variations in the base-emitter depletion region.

The data measured in the Rev_early and Fwd_early setups is interdependent. The extracted parameters are derived from the results of both measurements.

Parameters

The data from this measurement will be used to extract these parameters:

CJE	Base-emitter zero-bias capacitance
WBE	Portion of IBEI from Vbei, 1-WBE from Vbex
QB0	Model variable used in the Init_Parms macro to determine initial parameter values

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

From the early_aval DUT, select the **Rev_early** setup.

- The vs substrate voltage is usually set at approximately −3V to −1V.
- The vb base voltage is set to ground (0.0V).
- The ve emitter voltage should be set high enough that the base current just begins to turn negative: this value defines VBEmax, which is used in subsequent measurements. You may need to measure several times to determine the proper emitter voltage. Take care during these measurements not to break down the base-emitter junction: breakdown of this junction can permanently damage the device.
- The vc collector voltage should be around −650 mV. This results in a base-collector voltage of 0.65V, which should produce emitter current data with few thermal effects.
- The outputs are ib, ie, and is.

Measuring and Plotting

1 Make sure the device under test is reconnected in a measurement configuration.
2 From the Plots tab, select **Display All**. The plots for this setup are displayed.
3 Select **Measure/Simulate > Measure**.
4 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the early_aval Fwd_early Measurement

The forward early measurement models the effect of base-width modulation due to variations in the base-collector depletion region.

The Fwd_early measurement is performed over a range that includes data above avalanche breakdown for extraction of VEF and VER, as well as into the weak avalanche region for extraction of AVC1 and AVC2.

Because the Rev_early and Fwd_early setups are interdependent in the extraction of parameters, some of the measurement inputs are common between them.

Parameters

The data from this measurement will be used to extract these parameters:

VEF	Forward early voltage. Zero represents infinity
VER	Reverse early voltage. Zero represents infinity
AVC1	Base-collector weak avalanche parameter 1
AVC2	Base-collector weak avalanche parameter 2
CJE	Base-emitter zero-bias capacitance
CJEP	Base-collector extrinsic zero-bias capacitance

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

Select the **Fwd_early** setup.

- The vs substrate voltage is usually set at approximately −3V to −1V.
- The vb base voltage is set to ground (0.0V).
- The vc collector voltage should be set high enough that the base current just begins to turn negative: this value defines VBCmax, which is used in subsequent measurements. You may need to measure several times to determine the proper collector voltage.

- The ve emitter voltage should be around −650 mV. This results in a base-emitter voltage of 0.65V, which should produce collector current data with few thermal effects.

- The outputs are ic, collector current; ib, base current; and is, substrate current.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Ic_Vc plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Making the gummel Forward Measurement

This procedure characterizes the operating range of the base and collector currents with the transistor in the forward active mode. It measures Ic and Ib with respect to Vbe. The base-emitter voltage is swept across a range large enough that both low-level and high-level effects occur. Some later measurements are dependent on the results of the forward gummel measurement.

Parameters

The data from this measurement will be used to extract these parameters:

IS	Transport saturation current
NF	Forward emission coefficient
IBEI	Ideal base-emitter saturation current
NEI	Ideal base-emitter emission coefficient
IBEN	Non-ideal base-emitter saturation current
NEN	Non-ideal base-emitter emission coefficient

Defining the DC Source/Monitor Instrument State

Follow the process you used in the Rc_active measurement. Use the same settings, except set Integ Time to L to reduce trace noise at low voltage levels.

The Inputs and Outputs

From the gummel DUT, select the **Forward** setup.

- The vs substrate voltage is usually set at approximately −3V to −1V.
- The vb base voltage is set to ground (0.0V).
- The vc collector voltage is also set to ground (0.0V), so that the reverse junction (base-collector) is unbiased and will not affect the results.
- The ve emitter voltage is set to sweep from approximately −0.4V to −1.2V. The precise voltage values may vary. The intent is to measure low enough that the non-ideal base current effects are visible, and high enough in collector current to observe the compression due to high-level injection and/or quasi-saturation.
- The outputs are ic, collector current; ib, base current; and is, substrate current.

Measuring and Plotting

1 From the Plots tab, select **Display All**. Several plots are displayed.

2 Select **Measure/Simulate > Measure**. On the Ic_Vbe plot, the measured data should resemble the illustration. The vertical line on the trace is caused by the extraction algorithm, and will be explained in the extraction section later in this chapter.

Figure 168   Example Plotted gummel Forward Ic_Vbe Data

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Making the gummel Reverse Measurement

This procedure measures Ic and Ib with respect to Vbc. The base-collector voltage is swept across a range approximately the same as the base-emitter voltage range in the forward gummel measurement.

Parameters

The data from this measurement will be used to extract these parameters:

NR	Reverse emission coefficient
IBCI	Ideal base-collector saturation current
NCI	Ideal base-collector emission coefficient
IBCN	Non-ideal base-collector saturation current
NCN	Non-ideal base-collector emission coefficient
ISP	Parasitic transport saturation current

NFP	Parasitic forward emission coefficient
IKR	Reverse knee current. Zero represents infinity

Defining the DC Source/Monitor Instrument State

Follow the process you used in the Rc_active measurement. Use the same settings, except set Integ Time to L to reduce trace noise at low voltage levels.

The Inputs and Outputs

From the gummel DUT, select the **Reverse** setup.

- The vs substrate voltage is usually set at approximately −3V to −1V.
- The vb base voltage is set to ground (0.0V).
- The ve emitter voltage is also set to ground (0.0V), so that the reverse junction (base-emitter) is unbiased and will not affect the results.
- The vc collector voltage is set to sweep from approximately −0.4V to −1.2V. The precise voltage values may vary. The intent is to measure low enough that the non-ideal base current effects are visible, and high enough in emitter current to observe the compression due to high-level injection and/or quasi-saturation.
- The outputs are ie, emitter current; ib, base current; and is, substrate current.

Measuring and Plotting

1 From the Plots tab, select **Display All**. Several plots are displayed.
2 Select **Measure/Simulate > Measure**. The measured data in the Ie_Vbc and Ib_Vbc plots should resemble the illustrations. The vertical lines are caused by the extraction algorithms, and will be explained in the extraction section later.
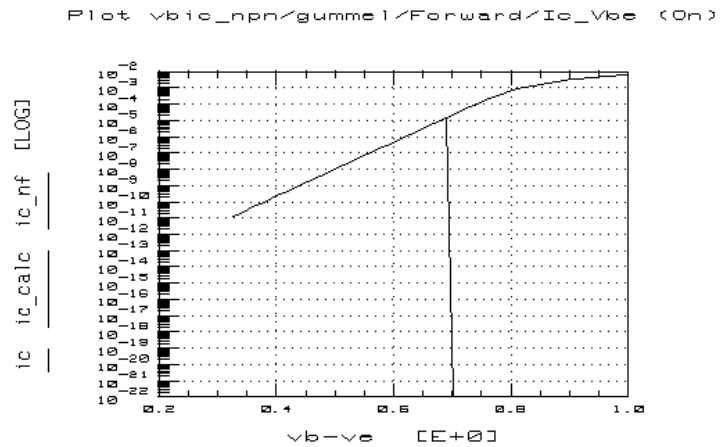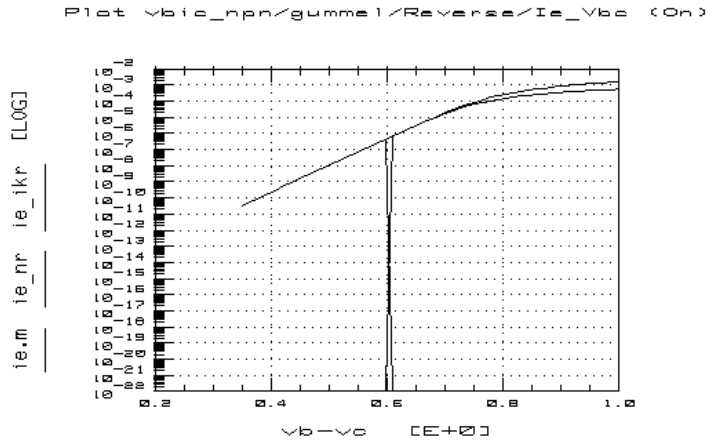
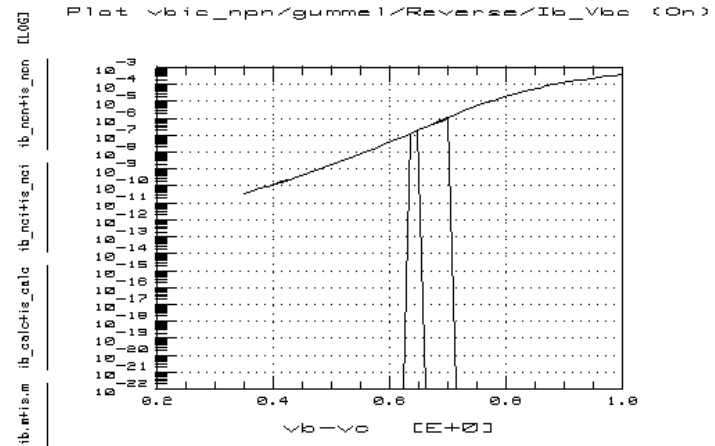**Figure 169**   Example Plotted gummel Reverse Ie_Vbc Data

Figure 170   Example Plotted gummel Reverse Ib_Vbc Data

Making the output Ic_Vce_vb Measurement

This procedure measures the device over a bias range where it exhibits both quasi-saturation behavior (the top one or two curves) and non-quasi-saturation behavior (the bottom one or two curves), while keeping Vbc < VBCmax where VBCmax defines the onset of weak avalanche breakdown. The data is used to extract the parameters related to high-level injection and quasi-saturation without avalanche effects.

Parameters

The data from this measurement will be used to extract these parameters:

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

From the output DUT, select the **Ic_Vce_vb** setup.

- The vs substrate voltage is usually set at −3V to −1V.
- The ve emitter voltage is set to ground (0.0V).
- The vb base voltage is set to measure at three points across its normal operating range.

- The vc collector voltage is swept from 0V to a value large enough that the output characteristics exhibit sufficient quasi-saturation and/or high-level injection without avalanche effects.

- The outputs are ic, collector current; vb, base voltage; and is, substrate current.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Ic_Vce plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Instrument States for CW S-Parameter Measurements

This information is included here because the first setups to use a CW frequency measurement are the setups in the quasi_sat DUT.

The instrument states for a CW S-parameter measurement are similar to those for a swept measurement, with a few important exceptions, listed below.

You will need to set the instrument states separately for *each* CW frequency measurement. However, use the same settings for all the CW measurement setups (except that Integ Time can differ among setups).

NOTE   Frequencies are set in the setup Inputs, therefore they do not affect instrument states even if you choose to use different CW frequencies for different measurements, except for the Cal Set No where they are stored.

1 The Agilent 4142 instrument state is the same as for the DC and swept S-parameter measurements.

2 Set Use User Sweep to Yes, as CW is a nonstandard sweep.

3 (Be sure to set Port 1 Atten and Port 2 Atten and Source Power to the same levels as you used in the network analyzer calibration.)

4 Set Use Fast CW to No.

5 Use the Cal Set No where you stored the CW calibration for the quasi_sat measurements. This must be different from the cal set used by the swept broadband calibration, and from any CW calibration you make at a different frequency for any different measurement.

6 If necessary, refer to the Agilent 8510C *Operating and Programming Manual* for more detail.

Making the quasi_sat QuasiSat_ac Measurement

This procedure measures the device in all of its regions: saturated, quasi-saturated, and cut off. It uses as few measurement points as possible, in order to speed up the subsequent optimization. This measures the device with a network analyzer at a single relatively low CW measurement frequency.

Parameters

The data from this measurement will be used to extract these parameters:

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity.
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

Defining the Instrument States

Follow the process described in "Instrument States for CW S-Parameter Measurements" on page 504.

The Inputs and Outputs

Select the **QuasiSat_ac** setup.

• The vs substrate voltage is set to ground (0.0V).

• The ve emitter voltage is set to ground (0.0V).

• The Sweep Type for the vb base voltage is set to LIST. Three values are defined, at low, medium, and high levels of the normal base operating range.

• The Sweep Type for vc collector voltage is set to LIST. The values are selected based on the following criteria:

  • Two data points in saturation: approximately 0V and just before the onset of quasi-saturation.

  • One data point at the transition knee between saturation and quasi- saturation.

  • Two data points in quasi-saturation: just past the onset of quasi-saturation and just before the onset of the active region.

  • One data point at the transition knee between quasi-saturation and the active region.

  • Two data points in the active region: one just past quasi-saturation and one before the onset of avalanche effects.

• The freq Sweep Type is set to CON for a single CW frequency. The value is typically in the range of approximately 100 to 200 MHz. This is high enough for adequate measurement resolution, yet low enough to avoid series parasitic effects.

• The output is S-parameters (s).

Measuring and Plotting

1 From the Plots tab, select **Display All**. Several plots are displayed.

2 Select **Measure/Simulate > Measure**. The plots displayed have already been converted to Y-parameters by one of the extraction routines.

3 Save the measured data using **File > Save As** and a filename with the suffix *.set*.

Making the quasi_sat QuasiSat_dc Measurement

This procedure makes a DC measurement of the device using the same bias values as the QuasiSat_ac measurement. The resulting data is used with the QuasiSat_ac data to optimize the relevant parameters.

Parameters

The data from this measurement will be used to extract the following parameters (the same parameters as the data from QuasiSat_ac):

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity.
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

Defining the DC Source/Monitor Instrument State

Follow the same process you used in the Rc_active measurement. Use the same settings.

The Inputs and Outputs

Select the **QuasiSat_dc** setup.

- The vs substrate voltage is set to ground (0.0V).
- The ve emitter voltage is set to ground (0.0V).
- The Sweep Type for the vb base voltage is set to LIST. Three values are defined, at low, medium, and high levels of the normal base operating range.
- The Sweep Type for vc collector voltage is set to LIST. The values are selected based on the following criteria:
  - Two data points in saturation: approximately 0V and just before the onset of quasi-saturation.

- One data point at the transition knee between saturation and quasi- saturation.

- Two data points in quasi-saturation: just past the onset of quasi-saturation and just before the onset of the active region.

- One data point at the transition knee between quasi-saturation and the active region.

- Two data points in the active region: one just past quasi-saturation and one before the onset of avalanche effects.

- The output is ic, collector current.

Measuring and Plotting

1 From the Plots tab, select **Display Plot**. The Ic plot is displayed.

2 Select **Measure/Simulate > Measure**.

3 Save the file using **File > Save As** and a filename with the suffix *.set*.

Making the quasi_sat Ypads Measurement

This setup is used to measure an open dummy structure. With the device disconnected, S-parameters are measured at the same CW frequency as in the QuasiSat_ac measurement. The data is used to de-embed the device S-parameters.

Parameters

No parameters are extracted from this data.

Defining the Instrument States

Follow the same process you used for the QuasiSat_ac measurement, described in "Instrument States for CW S-Parameter Measurements" on page 504.

The Inputs and Outputs

1 Remove the test fixture from the measurement setup, or lift the probes of a probe station to present an open circuit.

2 In the quasi_sat DUT select the **Ypads** setup.

- No voltages are defined or required.
- The CW frequency defined in freq must be the same as you used in QuasiSat_ac.
- The output is S-parameters (s).

Measuring

1 Remove the device under test from the fixture, or lift the probes of a probe station to present an open circuit.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the sparm_cap Cbe Measurement

The sparm_cap DUT measurements are similar to the cap DUT measurements, except that network analyzer S-parameter measurements are used instead of CV or LCR meter measurements. This can provide greater accuracy at higher frequencies.

NOTE

You need only do *one* set of capacitance measurements, using either the cap setups and a CV meter, or the sparm_cap setups and a network analyzer. You may find it more convenient to use the network analyzer, since it must be calibrated for other measurements anyway.

To effectively extract all three junction capacitances, and especially the substrate-collector capacitance, the substrate should be brought up to the top surface of the device and connected to a probeable pad, in such a way that the emitter and substrate can be RF-shorted through the probe. This provides a clean, clearly-defined collector-substrate path to the network analyzer ground, resulting in more accurate S-parameter measurements. If all the parasitic capacitances are

properly de-embedded, any capacitance between the collector and the RF ground will be completely due to the substrate capacitance.

The sparm_cap Cbe setup measures the base-emitter junction capacitance.

Parameters

The data from this measurement will be used to extract these parameters:

CJE	Base-emitter zero-bias capacitance
PE	Base-emitter built-in potential
ME	Base-emitter grading coefficient

Defining the Instrument States

Follow the same process you used for the QuasiSat_ac measurement, described in "Instrument States for CW S-Parameter Measurements" on page 504. Use the same settings, except if you used a different frequency be sure to set the correct number for the cal set where the calibration is stored.

The Inputs and Outputs

From the sparm_cap DUT, select the **Cbe** setup.

- The vb values are set to sweep the base voltage from approximately VBEmax to 0.5V.
- The vc collector voltage is set to ground (0.0V).
- The vs substrate voltage is set to ground (0.0V).
- The ve emitter voltage is set to ground (0.0V).

- The measurement frequency freq value is a balance between two constraints: the lower the frequency the smaller the series parasitics; at the same time, the higher the frequency the smaller the capacitance value the network analyzer can resolve. Typically the frequency should be in the range of approximately 100 to 200 MHz. Be sure you have stored a network analyzer CW calibration at the frequency you choose.
- The output is S-parameters (s).

Measuring and Plotting

1 Make sure the device under test is reconnected in a measurement configuration.

2 From the Plots tab, select **Display Plot**. The Cbe_Vbe plot is displayed.

3 Select **Measure/Simulate > Measure**.

4 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the sparm_cap Cbc_Csc Measurement

This procedure measures the base-collector and substrate-collector junction capacitances with the network analyzer.

Parameters

The data from this measurement will be used to extract these parameters:

CJC	Base-collector intrinsic zero-bias capacitance
PC	Base-collector built-in potential
MC	Base-collector grading coefficient
CJCP	Substrate-collector zero-bias capacitance
PS	Substrate-collector built-in potential
MS	Substrate-collector grading coefficient

Defining the Instrument States

Follow the same process you used in the sparm_cap Cbe measurement. Use the same settings.

The Inputs and Outputs

Select the **Cbc_Csc** setup.

- The vc values are set to sweep the collector voltage from approximately −0.5V to VBCmax (7.5V).
- The vb base voltage is set to ground (0.0V).
- The vs substrate voltage is set to ground (0.0V).
- The ve emitter voltage is set to ground (0.0V).
- The freq input must be set to the same CW frequency as in the Cbe measurement.
- The output is S-parameters (s).

Measuring and Plotting

1 From the Plots tab, select **Display All**. Two plots are displayed.
2 Select **Measure/Simulate > Measure**.
3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the sparm_cap Cpads Measurement

This setup is used to measure an open dummy structure with the network analyzer. With the device disconnected, S-parameters are measured at the same CW frequency as in the other sparm_cap measurements. The data is used to de-embed the device S-parameters.

Parameters

No parameters are extracted from this data.

Defining the Instrument States

Follow the same process you used in the sparm_cap Cbe measurement. Use the same settings.

The Inputs and Outputs

From the sparm_cap DUT, select the **Cpads** setup.

- No voltages are defined or required.
- The CW frequency defined in freq must be the same as you used in the other sparm_cap measurements.
- The output is S-parameters (s).

Measuring

1 Remove the test fixture from the measurement setup, or lift the probes of a probe station to present an open circuit.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the delay Ftvsic Measurement

This procedure determines the Ft of the device from bias-dependent S-parameters measured with the device in the common-emitter configuration. The S-parameters are measured at a single frequency as a function of Vbe and Vce. Typically two sets of curves are generated, one for each collector voltage value. The base-emitter voltage is swept over a very broad range.

Parameters

The data from this measurement will be used to extract these parameters:

TF	Forward transit time
XTF	Coefficient of TF bias dependence
ITF	Coefficient of TF dependence on Ic
VTF	Coefficient of TF dependence on Vbc
QTF	Variation of TF with base-width modulation

Defining the Instrument States

Follow the same process you used for the QuasiSat_ac measurement, described in "Instrument States for CW S-Parameter Measurements" on page 504. Use the same settings, except if you are using a different frequency be sure to set the correct number for the cal set where the calibration is stored.

The Inputs and Outputs

1 Make sure the device under test is reconnected in a measurement configuration.

2 From the delay DUT, select the **Ftvsic** setup.

   • The ve emitter voltage is set to ground (0.0V).

   • The vs substrate voltage is set to ground (0.0V).

   • The vb values are set to sweep the base voltage from a value where the depletion charge dominates (approximately 0.65V), to a value large enough that the decrease in Ft due to high-level injection effects is present (approximately 0.9V).

   • The vc collector voltage is usually swept from 1V to approximately 3–5V. If the collector voltage values are set too high, thermal problems begin to dominate the measurements and may cause thermal runaway. It is not uncommon for thermal runaway to occur even at medium voltages.

   • The freq frequency should be a value in the 20 dB/decade region, between F3dB and Ft. It must be a calibrated frequency.

   • The outputs are s, S-parameters; and ic, collector current.

Measuring and Plotting

1 Make sure the device under test is reconnected in a measurement configuration.

2 From the Plots tab, select **Display All**. Several plots are displayed.

3 Select **Measure/Simulate > Measure**.

4 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the delay Forward_Tau Measurement

This procedure measures the device S-parameters, which will be converted (after de-embedding) to H-parameters in the extraction process. Transforms will separate the H-parameter data to isolate the regions where the effects of specific parameters are greatest. This information is detailed in the parameter extraction section later in this chapter.

The measurement uses as few data points as possible to speed up the subsequent optimization. The input values for the base voltage and collector voltage are set in stepped mode.

It may be necessary to modify the measurement input bias values after the extraction results are seen, then perform the measurement and extraction again.

Parameters

The data from this measurement will be used to extract these parameters:

TF	Forward transit time
XTF	Coefficient of TF bias dependence
ITF	Coefficient of TF dependence on Ic
VTF	Coefficient of TF dependence on Vbc
QTF	Variation of TF with base-width modulation
QCO	Epi charge parameter

Defining the Instrument States

Follow the same process you used in the Ftvsic measurement. Use the same settings.

The Inputs and Outputs

Select the **Forward_Tau** setup.

- The vs substrate voltage is set to ground (0.0V).

- The ve emitter voltage is set to ground (0.0V).

- The freq value should be a frequency approximately 100 to 200 MHz. This is high enough for adequate measurement resolution, yet low enough to avoid series parasitic effects.

- The Sweep Type for the vb base voltage is set to LIST. A few measurement values of interest should be set, between approximately 0.65V and 0.9V. These correspond to the sweep range of the base voltage in the Ftvsic measurement.

- The Sweep Type for the vc collector voltage is set to LIST. Values are set to step from approximately 1V to approximately 3–5V, in as few as two steps. The values correspond to the sweep range of the collector voltage in the Ftvsic measurement.

- The outputs are S-parameters (s).

Measuring and Plotting

The plots are of H-parameters, and will probably be of more interest in the extraction process. However, the illustration shows the plot of H21 magnitude.
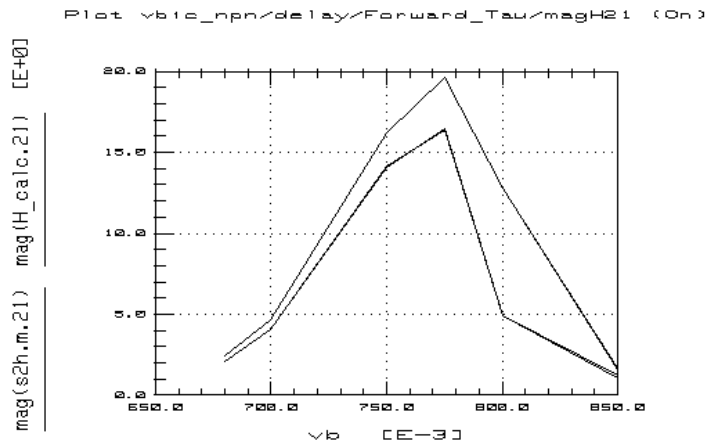
Figure 171   Example Plotted Forward_Tau magH21 Data

1 Select **Measure/Simulate > Measure**.

2 Save the data using **File > Save As** and a filename with the suffix *.set*.

Making the delay Reverse_Tau Measurement

This procedure measures small signal parameters of the device with the base-collector junction forward biased. With this bias, the effects of the base-collector diffusion capacitance, the epilayer charge, and the forward knee current of the parasitic PNP are all large enough to be discernible from the measured small signal data using optimization.

Parameters

The data from this measurement will be used to extract these parameters:

TR   Reverse transit time

QCO   Epi charge parameter

IKP   Parasitic knee current. Zero represents
                    infinity.

Defining the Instrument States

Follow the same process you used in the Ftvsic measurement.
Use the same settings.

The Inputs and Outputs

Select the **Reverse_Tau** setup.

- The vs substrate voltage is set to ground (0.0V).
- The ve emitter voltage is set to ground (0.0V).
- The vc collector voltage is set to ground (0.0V).
- The vb base voltage is set to sweep over a range from low to high, for example from 0.65V to 0.75V, using three to five measurement points to cover the range.
- freq is set to the same value as in Forward_Tau.
- The outputs are S-parameters (s).

Measuring and Plotting

1 Select **Measure/Simulate > Measure**.
2 Save the data using **File > Save As** and a filename with the suffix .*set*.

Making the delay Ypads Measurement

This setup is used to measure an open dummy structure. With the device disconnected, S-parameters are measured at the same CW frequency as in the other delay measurements. The data is used to de-embed the device S-parameters.

Parameters

No parameters are extracted from this data.

Defining the Instrument States

Follow the same process you used in the Ftvsic measurement. Use the same settings.

The Inputs and Outputs

From the delay DUT, select the **Ypads** setup.

- No voltages are defined or required.
- The CW frequency defined in freq must be the same as you used in the other delay measurements.
- The output is S-parameters (s).

Measuring

1 Remove the test fixture from the measurement setup, or lift the probes of a probe station to present an open circuit.

2 Select **Measure/Simulate > Measure**.

3 Save the data using **File > Save As** and a filename with the suffix .*set*.

Parameter Extractions

When the measurements are finished and you have a complete set of measured data for a particular device, you can begin the extractions.

Displaying the Parameters Table

1 Click the Model Parameters tab. The table of model parameters is displayed, with their current values. A portion of the table is illustrated.

Figure 172   Top Portion of the Model Parameters Table

The Parameters table lists all the parameters that will be extracted. It initially displays default values. As you perform the extraction macros, the default values change to extracted values. Throughout the extraction process, you can use the Parameters table to read extracted values.

2 To save the Parameters table after your device parameters are extracted, select **File > Save As**.

3 In the dialog box, select the File Type **.mps** for model parameter set.

4 Type a filename or use the default name in the File Name box.

5 Click **OK**.

You can now recall the same set of device parameters any time you wish.

Device Input Variable Values

Before the model is run, it is necessary to set values for some of the model variables. Model variables are used in different places throughout the model. Other variables that will be set later in the procedures are DUT variables, which apply to all the setups within a particular DUT, and setup variables, which apply to one particular setup.

Also, some of these variables will be set using the Init_Parms macro.

• To open the model variable table, click the **Model Variables** tab.

Name	Value
POLARITY	NPN
SIMULATOR	mns
TEMP	27.00
TNOM	27
TAMB	27
VBIC_ERROR_TOL	1e-12
VBIC_AUTO_RANGE	0
VBIC_AUTO_SMOOTH	0
qb0	3.45937E-13
SPARM_CAP	0
PLOT	0
VBIC_NUM_ITER	500
MNS_OPTIONS	maxdeltav=0.01
Nepi	1E+15
Na	5E+17
He	0.0004
Le	0.001
Wepi	0.0005
nb	2
ni	1.4E+10
VBIC_AUTO_LIMIT	0.9

Figure 173   The Model Variable Table

You need not set most of the values in the variable table. Some of these (for example *SIMULATOR: mns*) are default values that should not be changed. Some will be set automatically by the extraction macros.

VBIC_ERROR_TOL   The default is 1E–6. This is used by the internal DC and AC solvers to set the minimum allowable solver error. The smaller the value the more accurate the solution, but the slower the simulation.

VBIC_AUTO_RANGE   is set in the Init_Parms macro, and described there.

VBIC_AUTO_SMOOTH   is used in conjunction with *VBIC_AUTO_RANGE* (described under Init_Parms on the next page). If it is set to 1 the measured data determined by *VBIC_AUTO_RANGE* is smoothed before analysis. The smoothing algorithm returns the average of the current point, the next point, and the previous point in the data array. This is helpful if the measured data is noisy.

SPARM_CAP   is set in the Init_Parms macro, and described there.

PLOT   is set in the Init_Parms macro, and described there.

VBIC_NUM_ITER   The default is 500. This variable is used by the internal DC and AC solvers to set the maximum number of iterations the solvers will perform. If the number of iterations reaches the value set by this variable and the error is still larger than the number specified by *VBIC_ERROR_TOL*, the solver will stop and display an error message. Note that the smaller *VBIC_ERROR_TOL* is, the more likely the solver will be unable to converge to a stable solution.

VBIC_AUTO_LIMIT   The default is 0.9. The smaller the number, the tighter the limit used for auto ranging. If it is determined that the autoranging algorithm is incorporating too much high-level effect in its IS and NF extraction and simulation, the value of *VBIC_AUTO_LIMIT* can be made slightly larger than 0.9. Or if not enough log-linear region is being specified *VBIC_AUTO_LIMIT* can be made slightly smaller than 0.9.

Saving the Variable Table

You can store the variable table if you choose:

1 Select **File > Save As**.

2 In the dialog box, select the File Type *.vat* for variable table. Type in a filename or use the one in the File Name box. Select **OK**.

You can now recall the same set of variable values any time you wish to extract a similar device.

Initializing Device Parameters

Several device parameters need to be entered into the software before the extractions are begun. These include the length and width of the device, and certain process and geometry parameters. They also include certain model variables that were mentioned under "Device Input Variable Values" in the last few pages. More detailed descriptions are given here. (Infrequently used model variables are not set in the Init_Parms macro.)

The Init_Parms Macro

This macro is used to enter the required parameters. The more information you provide in this macro, the better the extraction will be.

1 Click the **macros** tab, and a list of macros is displayed.

2 In the Select Macro list, select **Init_Parms**. Then select **Execute**. A series of dialog boxes is displayed, asking you to input appropriate values for your device under test. The values you need to provide (and their units of measure) include:

   - epi dope (cm−3)
   - maximum base dope (cm−3)
   - emitter width (um)
   - emitter length (um)
   - epi layer thickness (um)
   - extraction temperature (degrees C)
   - number of base fingers (#)

3 You are then prompted to perform the capacitance measurements with network analysis.

   - If you choose y the model variable *SPARM_CAP* is set to 1, and all capacitance extractions are performed using network analyzer measurements from the sparm_cap DUT.
   - If you choose n the variable *SPARM_CAP* is set to 0 and all capacitance extractions are performed using the CV measurements from the cap DUT.

4 If you want the plots to be displayed during extraction, choose y. The model variable *PLOT* will be set to 1. If not, choose n and the variable *PLOT* will be set to 0.

5 If you want the extractions to use automatic data selection, choose y. The model variable *VBIC_AUTO_RANGE* will be set to 1. If not, choose n and it will be set to 0. When *VBIC_AUTO_RANGE* is set to 1, the autorange algorithm is turned on. This algorithm evaluates the measured data to determine the valid range for a given extraction transform, based on the parameter field specified in the transform. The auto range typically works by looking at the 1st and 2nd derivatives of a measured current. When the derivatives deviate from the value set by VBIC_AUTO_LIMIT, the output of the function is set to 0.

The Automated Macros

Macros automate the modeling procedure so that parameter extraction and optimization can be performed with minimal user intervention.

The fastest and easiest way to perform the extractions is to use the Do_all macro (described below). Otherwise, the macros can be executed separately, each extracting a subset of model parameters.

The macros call the transforms necessary to extract the relevant parameters. If the model variable *PLOT* is set to 1, the applicable plot or plots are displayed. If the model variable *VBIC_AUTO_RANGE* is set to 1, the extractions use automatic data selection.

To run a macro, select the macro name and **Execute**.

Init_Parms This macro sets an initial set of parameter values based on device-specific physical data. Note it is not essential that the device or process information be known exactly: however, setting initial values, even if estimated, improves the extraction process. This macro also sets the commonly-used model variables *SPARM_CAP*, *PLOT*, and *VBIC_AUTO_RANGE*.

Do_all   Executes a complete extraction of the VBIC parameters by calling the other macros in the appropriate order.

Cbe   Calls the transforms used to extract the base-emitter junction capacitance parameters. If the variable *SPARM_CAPS* = 1 the sparm_caps (network analyzer measurement) data is used. Otherwise the cap (CV meter measurement) data is used.

Cbc   Calls the transforms used to extract the base-collector junction capacitance parameters. If *SPARM_CAPS* = 1 the sparm_caps data is used. Otherwise the cap data is used.

Csc   Calls the transforms used to extract the substrate-collector junction capacitance parameters. If *SPARM_CAPS* = 1 the sparm_caps data is used. Otherwise the cap data is used.

Re   Calls the transforms used to extract the emitter series resistance parameter RE.

Rcx   Calls the transforms used to extract the extrinsic collector resistance parameter RCX.

Early   Calls the transforms used to extract VEF and VER, and partitions the total base-collector capacitance between the intrinsic (CJC) and extrinsic (CJEP) portions of the device.

Aval   Calls the transforms used to extract the avalanche parameters AVC1 and AVC2.

Forward   Calls the transforms used to extract the parameters IS, NF, IBEI, NEI, IBEN, and NEN.

Reverse   Calls the transforms used to extract the parameters NR, IBCI, NCI, IBCN, NCN, ISP, NFP, and IKR.

Rbb   Calls the transforms used to extract the parameters RBX, RBI, and WBE.

Misc Calls the transforms used to extract the parameters TR, QCO, and IKP.

QuasiSat Calls the transforms used to extract the parameters IKF, RCI, RCX, GAMM, VO, HRCF, and QCO.

Delay Calls the transforms used to extract the parameters TF, QTF, VTF, XTF, ITF, and QCO.

The Transforms

The individual transforms that are called by the macros are described in the following pages. They are grouped according to the setups in which they occur.

To see the transforms, select the setup of interest from the DUT/Setup panel, then click on the **Extract/Optimize** tab. A list of the transforms for that setup is displayed. Click on the name of a transform to see its function and description.

To run an individual transform (though you will seldom need to), select **Execute**.

Transforms in the Cbe Setup

The parameters extracted are:

CJE	Base-emitter zero-bias depletion capacitance
PE	Base-emitter built-in potential
ME	Base-emitter grading coefficient

The transforms are:

extr_CJE	The function is VBIC_cj0. The transform calculates the zero-bias depletion capacitance given vbe and cbe.m.
cbe_calc	The function is VBIC_cbe. The transform calculates the depletion capacitance vs bias given vbe.

opt_CJE_PE_ME Optimization. This optimizes the simulated
value cbe_calc to obtain the final value for
cbe.m. The optimization is performed over the
full range of the measurement. All three
parameters are optimized.

Transforms in the Cbc Setup

The parameters extracted are:

CJC	Base-collector intrinsic zero-bias depletion capacitance
PC	Base-collector built-in potential
MC	Base-collector grading coefficient

The transforms are:

extr_CJC	The function is VBIC_cj0. The transform calculates the zero-bias depletion capacitance given vbc and cbc.m.
cbc_calc	The function is VBIC_cbc. The transform calculates the depletion capacitance vs bias given vbc.
opt_CJC_PC_MC	Optimization. This optimizes the simulated value cbc_calc to obtain the final value for cbc.m. The optimization is performed over the full range of the measurement. All three parameters are optimized.

Transforms in the Csc Setup

The parameters extracted are:

CJCP	Substrate-collector zero-bias depletion capacitance
PS	Substrate-collector built-in potential

MS   Substrate-collector grading coefficient

The transforms are:

extr_CJCP   The function is VBIC_cj0. The transform
                  calculates the zero-bias depletion capacitance
                  given vsc and csc.m.

csc_calc   The function is VBIC_csc. The transform
                  calculates the depletion capacitance vs bias
                  given vsc.

opt_CJCP_PS_MS Optimization. This optimizes the simulated
                  value csc_calc to obtain the final value for
                  csc.m. The optimization is performed over
                  the full range of the measurement. All three
                  parameters are optimized.
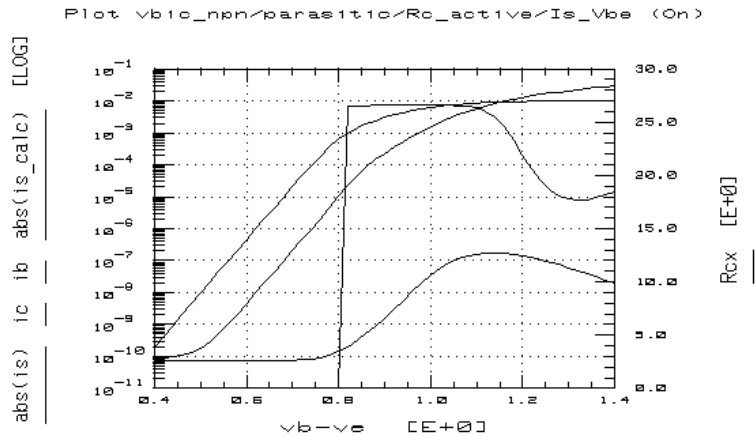
Transforms in the Rc_active Setup

The parameter extracted is:

RCX   Extrinsic collector resistance

The transforms are:

extr_RCX   This transform calculates RCX based on
                  Berkner's method. This uses the fact that the
                  substrate increases with an increasing voltage
                  drop across the constant portion of the
                  collector resistor when the device is in either
                  quasi-saturation or saturation. Because the
                  noise level of the substrate current is often
                  quite high, the dynamic range of this method
                  may not always be sufficient to resolve the
                  bias-independent collector resistance
                  adequately. An additional extraction is
                  performed in the Rc_flyback setup.

The vertical line on the trace is caused by the automatic data selection algorithm in the transform (called by the *VBIC_AUTO_RANGE* variable), which chops off the non-valid noisy data at the low end of the trace. RCX is the maximum point of the trace.

Figure 174   Plotted Results of the extr_RCX Transform

This method is described in detail in *A Survey of DC Methods for Determining the Series Resistance of Bipolar Transistors Including the New Delta ISub Method* (see Bibliography).

Transforms in the Re_flyback Setup

The parameter extracted is:

RE   Emitter resistance

The transforms are:

extr_RE   Calculates RE given ib and vc.m.

Transforms in the Rc_flyback Setup

The parameter extracted is :

RCX   Extrinsic collector resistance

The transforms are:

extr_RCX   Calculates RCX given ib and ve.m.

Transforms in the Rbb Setup

The parameters extracted are:

RBX   Extrinsic base resistance
RBI   Intrinsic base resistance
WBE   The portion of IBEI from Vbei, 1–WBE from
                 vbex.

The transforms are:

S_calc   The function is VBIC_ac_solver. This solves for
                 the S-parameters, given the four terminal
                 voltages.
deembed_S   This is a program that de-embeds the
                 S-parameters from the setup. It subtracts the
                 dummy structure data measured in the Ypads
                 setup from the S-parameters measured in this
                 Rbb setup: that is, it removes the parasitic
                 capacitance of the pads. The output of this
                 transform should be used for all subsequent
                 extractions and optimizations in this setup.

opt_RBX_RBI_
WBE

Optimization. This optimizes the de-embedded S-parameters to obtain the value s.m. The input to this transform is deembed_S.m, the S-parameter data de-embedded by the transform deembed_S. The optimization is performed over the full range of the measurement. All three parameters are optimized.

Transforms in the parasitic Ypads Setup

No parameters are extracted. The measured data is used to de-embed the device S-parameters.

The transforms are:

s2y

The function is TwoPort. The transform converts the measured S-parameters to Y-parameters.

c11

This is a program that takes the output from the transform s2y and extracts the input capacitance value based on an equivalent pi structure for the parasitic capacitance.

c12

This is a program that takes the output from the transform s2y and extracts the crossover capacitance value based on an equivalent pi structure for the parasitic capacitance.

c22

This is a program that takes the output from the transform s2y and extracts the output capacitance value based on an equivalent pi structure for the parasitic capacitance.

Transforms in the Rev_early Setup

The parameters extracted are:

CJE

Base-emitter zero-bias capacitance

WBE

Portion of IBEI from Vbei, 1−WBE from Vbex

QB0   Model variable used to determine initial parameter values

The transforms are:

ie_calc   The function is VBIC_dcv_solver. The transform calculates ie, given the four terminal voltages.

ib_calc   The function is VBIC_dcv_solver. The transform calculates ib, given the four terminal voltages.

extr_qb0   This is a program that calculates the initial value of QB0 based on a combination of measured data (ie.m and vbe) and extracted parameters (CJE and WBE). Therefore these parameters and variable must be determined first. The model variable QB0 is only used in the Init_Parms macro to determine initial parameter values.

Transforms in the Fwd_early Setup

The parameters extracted are:

VEF   Forward early voltage. Zero represents infinity

VER   Reverse early voltage. Zero represents infinity

AVC1   Base-collector weak avalanche parameter 1

AVC2   Base-collector weak avalanche parameter 2

CJC   Intrinsic base-collector zero-bias capacitance

CJEP   Base-collector extrinsic zero-bias capacitance

The transforms are:

ic_calc	The function is VBIC_dcv_solver. The transform calculates ic, given the four terminal voltages.
ib_calc	The function is VBIC_dcv_solver. The transform calculates ib, given the four terminal voltages.
extr_AVC1	The function is VBIC_avc. The transform calculates AVC1 based on the model parameter PC. The calculation is based on the original Mextram formulation described in (2) (Bibliography). The VBIC parameter AVC1 is equivalent to a constant in the Mextram model:

$$\frac{(AVC1 = A)}{b(1 - Pc)}$$

where:

a = 7.05E05 cm^{-1}b = 2.04E06 V/cm for PNP

b = 1.23E06 V/cm for NPN 1.58E06 cm^{-1}

Pc = b–c grading coefficient

extr_VEF_VER	The function is VBIC_vef_ver. The transform calculates the forward and reverse early voltages, given the collector, base, and emitter voltages in the forward and reverse modes, as well as the collector current in the forward mode and the emitter current in the reverse mode. This calculation is based on the method described in "Spice Early Model. McAndrew and Nagel. IEEE 1994 BCTM." on page 561.

If the model variable *VBIC_AUTO_RANGE* is turned on (set to 1), the transform requires an additional input, ib (base current). The base current is used to monitor avalanche breakdown: once avalanche breakdown begins all subsequent data is ignored by the transform.

The vertical line is caused by the automatic data selection algorithm called by VBIC_AUTO_RANGE. The data below the vertical line is invalid for the extraction.
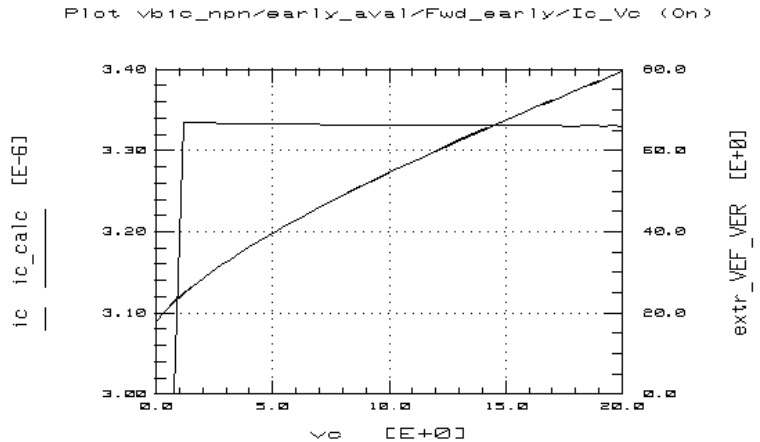

Figure 175   Plotted Fwd_early Data Showing Auto-Range Limits

opt_AVC1_AVC2 Optimization. This optimizes ib_calc to obtain the value ib.m. The optimization is performed over the full range of the measurement. The parameters optimized are the base-collector weak avalanche parameters AVC1 and AVC2.

VBIC_CJEP_CJC This is a program that partitions the total CJC into the intrinsic (CJC) and extrinsic (CJEP) capacitances, based on the 1–D intrinsic equality

$J = CJCint*VEF$.

Transforms in the gummel Forward Setup

The parameters extracted are:

IS	Transport saturation current
NF	Forward emission coefficient
IBEI	Ideal base-emitter saturation current
NEI	Ideal base-emitter emission coefficient
IBEN	Non-ideal base-emitter saturation current
NEN	Non-ideal base-emitter emission coefficient

The transforms are:

ic_calc	The function is VBIC_dcv_solver. The transform calculates ic, given the four terminal voltages.
ib_calc	The function is VBIC_dcv_solver. The transform calculates ib, given the four terminal voltages.
is_calc	The function is VBIC_dcv_solver. The transform calculates is, given the four terminal voltages.
ic_nf	The function is VBIC_fg_currents. The transform calculates ic over the region dominated by NF, given the four terminal voltages. This transform requires the measured inputs ic.m and ib.m. It also requires that the model variable *VBIC_AUTO_RANGE* is set to 1, and that the parameter NF is entered in the parameter field. Otherwise the simulation will be performed over the entire measurement range.

The vertical line on the trace is caused by the automatic data selection algorithm called by *VBIC_AUTO_RANGE*. The high-level injection effects above the vertical line are invalid for the extraction.
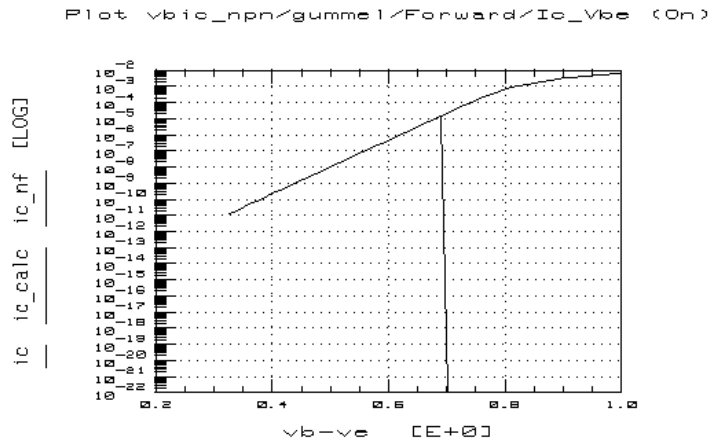
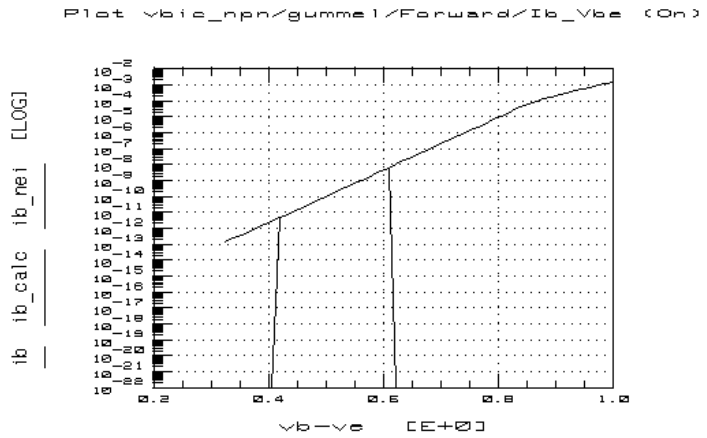**Figure 176**   Plotted ic Data Showing Auto Range Limits

Figure 177   Plotted ib Data Showing Auto Range Limits

ib_nei   The function is VBIC_fg_currents. The transform calculates ib over the region dominated by NEI, given the four terminal voltages. This transform requires the measured inputs ic.m and ib.m. It also requires that the model variable *VBIC_AUTO_RANGE* is set to 1, and that the parameter NEI is entered in the parameter field. Otherwise the simulation will be performed over the entire measurement range.

beta_nen   The function is VBIC_fg_currents. The transform calculates beta over the region dominated by NEN, given the four terminal voltages. This transform requires the measured inputs ic.m and ib.m. It also requires that the model variable *VBIC_AUTO_RANGE* is set to 1, and that the parameter NEN is entered in the parameter field. Otherwise the simulation will be performed over the entire measurement range.

extr_IS_NF   The function is VBIC_is_nf. The transform calculates the parameters IS and NF based on the method described in (3) (Bibliography). If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used for calculating IS and NF to the log-linear region of the collector current.

extr_IBEI_NEI   The function is VBIC_ibei_nei. The transform calculates the parameters IBEI and NEI. If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used: the limits are from above where the low-level effects dominate up to the onset of high-injection effects.

opt_IBEN_NEN   Optimization. This optimizes the simulated value beta_nen to obtain the value beta.m. The optimization is performed over the full range of the measurement.

opt_IS_NF   Optimization. This optimizes the simulated value log(ic_nf) to obtain the value log(ic.m). The optimization is performed over the full range of the measurement.

opt_IBEI_NEI   Optimization. This optimizes the simulated value log(ib_nei) to obtain the value log(ib.m). The optimization is over the full range of the measurement.

Transforms in the gummel Reverse Setup

The parameters extracted are:

NR   Reverse emission coefficient

IBCI   Ideal base-collector saturation current

NCI   Ideal base-collector emission coefficient

IBCN   Non-ideal base-collector saturation current

NCN   Non-ideal base-collector emission coefficient

ISP   Parasitic transport saturation current

NFP   Parasitic forward emission coefficient

IKR   Reverse knee current. Zero represents infinity.

The transforms are:

ie_calc   The function is VBIC_dcv_solver. The transform calculates ie, given the four terminal voltages.

ib_calc   The function is VBIC_dcv_solver. The transform calculates ib, given the four terminal voltages.

is_calc   The function is VBIC_dcv_solver. The transform calculates is, given the four terminal voltages.

|	
ie_nr, is_ncn, beta_ncn, is_isp, ib_nci, is_nci, ie_ikr, ib_ncn	These transforms are used in subsequent optimizations. They all require that the model variable *VBIC_AUTO_RANGE* is set to 1, and that the appropriate parameter is entered into the parameter field. They also require the measured inputs ie.m, is.m, and ib.m+is.m. Otherwise the simulations will be performed over the entire measurement range.
	In the IB field of these transforms it is usually preferable to enter the sum ib.m+is.m. The substrate current has an opposite polarity compared to the base current. Therefore the contribution of the substrate current is removed from the base current by adding the two quantities. The result is a better approximation of the true intrinsic base current. If the parasitic PNP had an infinite beta, the approximation would be exact.
	These transforms all use the function VBIC_rg_currents.
ie_nr	Calculates ie over the region dominated by NR, given the four terminal voltages. The vertical lines on the trace are caused by the automatic data selection algorithm called by *VBIC_AUTO_RANGE*. Only the high-level injection effects above the vertical lines are valid for the extraction.
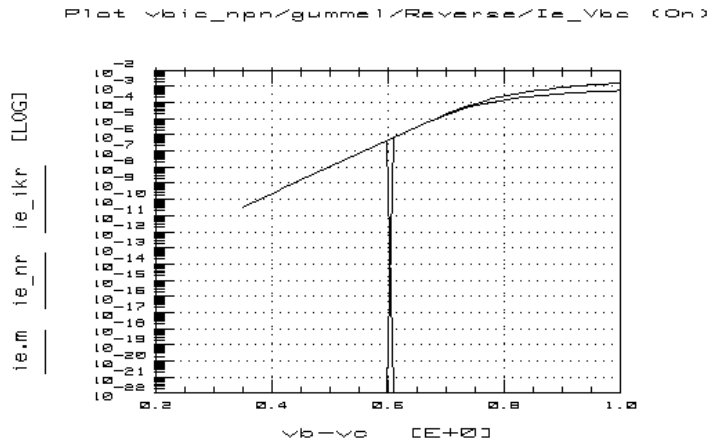
Figure 178   Plotted ie Data Showing Auto Range Limits

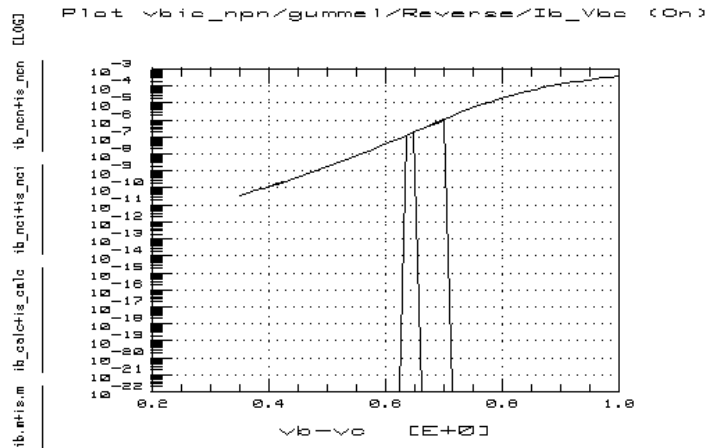|	
is_ncn	Calculates is over the region dominated by NCN, given the four terminal voltages.
beta_ncn	Calculates beta over the region dominated by NCN, given the four terminal voltages.
is_isp	Calculates is over the region dominated by ISP, given the four terminal voltages.
ib_nci	Calculates ib over the region dominated by NCI given the four terminal voltages.

Figure 179   Plotted ib Data Showing Auto Range Limits

is_nci	Calculates is over the region dominated by NCI, given the four terminal voltages.
ie_ikr	Calculates ie over the region dominated by IKR, given the four terminal voltages.
ib_ncn	Calculates ib over the region dominated by NCN, given the four terminal voltages.
extr_NR	The function is VBIC_nr. The transform calculates the parameter NR based on the method described in (3) (Bibliography). If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used to calculate NR to the log-linear region of the emitter current.
extr_IBCI_NCI	The function is VBIC_ibci_nci. The transform calculates the parameters IBCI and NCI. If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used: the limits are from above where the low-level effects dominate, up to the onset of high-injection effects.

extr_ISP_NFP   The function is VBIC_isp_nfp. The transform calculates the parameters ISP and NFP. If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used to the log-linear region of the substrate current.

extr_IKR   The function is VBIC_ikr. The transform calculates the parameter NR based on the method described in (3) (Bibliography). If *VBIC_AUTO_RANGE* is turned on, the transform automatically limits the data used to the log-linear region of the emitter current.

opt_IBCN_NCN   Optimization. This optimizes the simulated value beta_nen to obtain the value beta.m. The optimization is performed over the full range of the measurement.

opt_NR   Optimization. This optimizes the simulated value log(ie_nr) to obtain the value log(ie.m). The optimization is performed over the full range of the measurement.

opt_IBCI_NCI   Optimization. This optimizes the simulated value log(ib_nci+is_nci) to obtain the value log(ib.m+is.m). The optimization is performed over the full range of the measurement.

opt_ISP_NFP   Optimization. This optimizes the simulated value log(is_isp) to obtain the value log(is.m). The optimization is performed over the full range of the measurement.

opt_IKR   Optimization. This optimizes the simulated value log(ie_ikr) to obtain the value log(ie.m). The optimization is performed over the full range of the measurement.

beta   Equation. This calculates beta as ie/(ib + is). To a first order, this equation removes the effects of the parasitic PNP from the beta calculation.

Transforms in the Ic_Vce_Vb Setup

The parameters extracted are:

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity.
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

The same parameters are extracted in the QuasiSat_ac and QuasiSat_dc setups.

The transforms are:

ic_calc	The function is VBIC_dcv_solver. The transform calculates ic, given the four terminal voltages.
opt_ALL	Optimization. This optimizes the simulated value log(ic_calc) to obtain the value log(ic.m). The optimization is performed over the full range of the measurement. All the parameters are optimized (HRCF, VO, QCO, GAMM, IKF, RCI, and RCX).

Transforms in the QuasiSat_ac Setup

The parameters extracted are:

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity.
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

The same parameters are extracted in the Ic_Vce_Vb setup and the QuasiSat_dc setup.

The transforms are:

s2y	The function is TwoPort. This transform converts the de-embedded S-parameters into Y-parameters. The input to this transform is deembed_S.m, the S-parameters de-embedded by the transform deembed_S.
Y_calc	The function is VBIC_ac_solver. The transform solves for the Y-parameters, given the four terminal voltages.
deembed_S	This program de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the quasi_sat Ypads setup from the S-parameters measured in this QuasiSat_ac setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.
opt_IKF_RCI_RCX	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

log(QuasiSat_dc/ic_calc)	log(QuasiSat_dc/ic.m)
real(Y_calc.12)	real(s2y.m.12)
imag(Y_calc.12)	imag(s2y.m.12)
real(Y_calc.22)	real(s2y.m.22)
imag(Y_calc.22)	imag(s2y.m.22)

The optimization is performed over the full range of the measurement. The parameters optimized are IKF, RCI, and RCX.

opt_GAMM_QCOOptimization. This optimizes the following simulated values to obtain the corresponding listed final values:

log(QuasiSat_dc/ic_calc)	log(QuasiSat_dc/ic.m)
real(Y.calc.12)	real(s2y.m.12)
imag(Y_calc.12)	imag(s2y.m.12)
real(delay/Reverse_Tau_H/calc.21)	real(delay/Reverse_Tau/s2h.m.21)
imag(delay/Reverse_Tau/H_calc.21)	imag(delay/Reverse_Tau/s2h.m.21)

The optimization is performed over the full range of the measurement. The parameters optimized are GAMM and QCO.

opt_VO_HRCF Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

log(QuasiSat_dc/ic_calc)	log(QuasiSat_dc/ic.m)
real(Y_calc.22)	real(s2y.m.22)
imag(Y_calc.22)	imag(s2y.m.22)

The optimization is performed over the full range of the measurement. The parameters optimized are VO and HRCF.

opt_ALL Optimization. This optimizes the simulated value *ic_calc* to obtain the final value for *ic.m*. The optimization is performed over the full range of the measurement. All the parameters for this measurement are optimized.

Transforms in the QuasiSat_dc Setup

The parameters extracted are:

HRCF	High-current RC factor
VO	Epi drift saturation voltage
QCO	Epi charge parameter
GAMM	Epi doping parameter
IKF	Forward knee current. Zero represents infinity.
RCI	Intrinsic collector resistance
RCX	Extrinsic collector resistance

The same parameters are extracted in the Ic_Vce_Vb setup.

The transforms are:

ic_calc	The function is VBIC_dcv_solver. The transform calculates ic, given the four terminal voltages.
opt_ALL	Optimization. This optimizes the simulated value log(ic_calc) to obtain the final value log(ic.m). The optimization is performed over the full range of the measurement. All the parameters listed above are optimized.
init_QS	This is a program that performs an analytical calculation of an initial guess for the quasi-saturation parameters RCI, GAMM, VO, and QCO. This is based on the equations of Kull (1985). The calculation uses the following variables:

Nepi	Epi doping concentration.
Wepi	Epi layer thickness.
Le	Emitter length.
He	Emitter width.

Transforms in the quasi_sat Ypads Setup

No parameters are extracted. The measured data is used to de-embed the device S-parameters.

The transforms are:

s2y	The function is TwoPort. The transform converts the measured S-parameters to Y-parameters.
c11	This is a program that takes the output from the transform s2y and extracts the input capacitance value based on an equivalent pi structure for the parasitic capacitance.
c12	This is a program that takes the output from the transform s2y and extracts the crossover capacitance value based on an equivalent pi structure for the parasitic capacitance.
c22	This is a program that takes the output from the transform s2y and extracts the output capacitance value based on an equivalent pi structure for the parasitic capacitance.

Transforms in the sparm_cap Cbe Setup

The parameters extracted are:

CJE	Base-emitter zero-bias capacitance
PE	Base-emitter built-in potential
ME	Base-emitter grading coefficient

The transforms are:

deembed_S   This is a program that de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the sparm_cap Cpads setup from the S-parameters measured in this Cbe setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.

cbe   The function is BJTCV_stoc. The transform calculates the base-emitter capacitance, given the measurement frequency and the S-parameters. The input to this transform is the S-parameter data de-embedded by the transform deembed_S. The optimization is performed over the full range of the measurement.

extr_CJE   The function is VBIC_cj0. The transform calculates the zero-bias depletion capacitance CJE, given vbe and cbe.m.

cbe_calc   The function is VBIC_cbe. The transform calculates the depletion capacitance vs bias, given vbe.

opt_CJE_PE_ME Optimization. This optimizes the simulated value cbe_calc to obtain the final value cbe.m. The optimization is performed over the full range of the measurement.

Transforms in the sparm_cap Cbc_Csc Setup

The parameters extracted are:

CJC   Base-collector intrinsic zero-bias capacitance

PC   Base-collector built-in potential

MC   Base-collector grading coefficient

CJCP   Substrate-collector zero-bias capacitance

PS   Substrate-collector built-in potential

MS Substrate-collector grading coefficient

The transforms are:

deembed_S This is a program that de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the sparm_cap Cpads setup from the S-parameters measured in this Cbc_Csc setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.

cbc The function is BJTCV_stoc. The transform calculates the base-collector capacitance, given the measurement frequency and the S-parameters. The input to this transform is the S-parameter data de-embedded by the transform deembed_S. The optimization is performed over the full range of the measurement.

csc The function is BJTCV_stoc. The transform calculates the substrate-collector capacitance, given the measurement frequency and the S-parameters. The input to this transform is the S-parameter data de-embedded by the transform deembed_S. The optimization is performed over the full range of the measurement.

extr_CJC The function is VBIC_cj0. The transform calculates the zero-bias depletion capacitance, given vbc and cbc.m.

cbc_calc The function is VBIC_cbc. The transform calculates the depletion capacitance vs bias, given vbc.

opt_CJC_PC_MC   Optimization. This optimizes the simulated value cbc_calc to obtain the final value cbc.m. The optimization is performed over the full range of the measurement.

extr_CJCP   The function is VBIC_cj0. The transform calculates the zero-bias depletion capacitance CJCP, given vsc and csc.m.

csc_calc   The function is VBIC_csc. The transform calculates the depletion capacitance vs bias, given vsc.

opt_CJCP_PS_MS Optimization. This optimizes the simulated value csc_calc to obtain the final value csc.m. The optimization is performed over the full range of the measurement.

Transforms in the sparm_cap Cpads Setup

No parameters are extracted. The measured data is used to de-embed the device S-parameters.

The transforms are:

s2y   The function is TwoPort. The transform converts the measured S-parameters to Y-parameters.

c11   This is a program that takes the output from the transform s2y and extracts the input capacitance value based on an equivalent pi structure for the parasitic capacitance.

c12   This is a program that takes the output from the transform s2y and extracts the crossover capacitance value based on an equivalent pi structure for the parasitic capacitance.

c22   This is a program that takes the output from the transform s2y and extracts the output capacitance value based on an equivalent pi structure for the parasitic capacitance.

Transforms in the Ftvsic Setup

In this setup, the Ft of the device is determined from bias-dependent S-parameters measured in the common-emitter configuration. The S-parameters are measured at a single frequency as a function of Vbe and Vce.

The parameters extracted are:

TF	Forward transit time
XTF	Coefficient of TF bias dependence
ITF	Coefficient of TF dependence on Ic
VTF	Coefficient of TF dependence on Vbc
QTF	Variation of TF with base-width modulation
QCO	Epi charge parameter

The transforms are:

ic_calc	The function is VBIC_dcv_solver. The transform calculates ic, given the four terminal voltages.
s2h	The function is TwoPort. The transform converts the de-embedded S-parameters to H-parameters.
Ft_calc	The function is VBIC_ac_solver. The transform solves for the Ft of the device, given the four terminal voltages.
H_calc	The function is VBIC_ac_solver. The transform solves for the H-parameters, given the four terminal voltages.
deembed_S	This is a program that de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the delay Ypads setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.

Ft	Equation. This calculates Ft based on the measured data freq*mag(H21). The equation assumes operation in the single-pole roll-off region (20 db/decade).
init_Delay	This is a program that performs an analytical calculation of initial guesses for the delay parameters TF, XTF, ITF, VTF, and QTF.

Transforms in the Forward_Tau Setup

The parameters extracted are:

TF	Forward transit time
XTF	Coefficient of TF bias dependence
ITF	Coefficient of TF dependence on Ic
VTF	Coefficient of TF dependence on Vbc
QTF	Variation of TF with base-width modulation
QCO	Epi charge parameter

This setup uses the standard H21 measurements, but divides the measurement data into the specific regions where the effect (sensitivity) of different parameters is greatest. The regions for optimization are as follows:

TF	High Vce; Vbe range from low to just before peak Ft
ITF	High Vce; Vbe range from just past peak Ft to a high value
XTF	Any value Vce; Vbe from just after peak Ft to a high value
VTF	Any value Vce; Vbe from just after peak Ft to a high value
QTF	Any value Vce; Vbe medium (including peak Ft value)

For this purpose, several PEL transforms are used within the main transforms: TF_Hparm, QTF_Hparm, ITF_Hparm, VTF_Hparm, XTF_Hparm, and QCO_Hparm. As few data points as possible are used to speed up optimization. Each optimization point uses the approximation of the last to assist in converging to a minimum value. The bias values and subsequently the i, k, and l specified values in the transforms will probably need to be modified for the particular device being characterized.

The transforms are:

s2h	The function is TwoPort. The transform converts the de-embedded S-parameters to H-parameters.
H_calc	The function is VBIC_ac_solver. The transform solves for the H-parameters, given the four terminal voltages.
deembed_S	This is a program that de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the delay Ypads setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.
opt_TF_QTF	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

mag(TF_Hparm.s.21)	mag(TF_Hparm.m.21)
ph(TF_Hparm.s.21)	ph(TF_Hparm.m.21)
mag(QTF_Hparm.s.21)	mag(QTF_Hparm.m.21)
ph(QTF_Hparm.s.21)	ph(QTF_Hparm.m.21)

opt_TF	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values.

mag(TF_Hparm.s.21)	mag(TF_Hparm.m.21)
ph(TF_Hparm.s.21)	ph(TF_Hparm.m.21)

opt_QTF_XTF_VTF_ITF_QCO	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

mag(QTF_Hparm.s.21)	mag(QTF_Hparm.m.21)
ph(QTF_Hparm.s.21)	ph(QTF_Hparm.m.21)
mag(XTF_Hparm.s.21)	mag(XTF_Hparm.m.21
ph(XTF_Hparm.s.21)	ph(XTF_Hparm.m.21)
mag(VTF_Hparm.s.21)	mag(VTF_Hparm.m.21)
ph(VTF_Hparm.s.21)	ph(VTF_Hparm.m.21)
mag(ITF_Hparm.s.21)	mag(ITF_Hparm.m.21)
ph(ITF_Hparm.s.21)	ph(ITF_Hparm.m.21)
mag(QCO_Hparm.s.21)	mag(QCO_Hparm.m.21)
ph(QCO_Hparm.s.21)	ph(QCO_Hparm.m.21)

Transforms in the Reverse_Tau Setup

The purpose of this setup is to measure the small signal parameters of the device with the base-collector junction forward biased. Under this bias, the effects of the base-collector diffusion capacitance (TR), epilayer charge (QCO), and forward knee current of the parasitic PNP (IKP) are all large enough to be discernible from the measured small signal data through optimization.

The parameters extracted are:

TR	Base-collector diffusion capacitance (reverse transit time)
QCO	Epi charge parameter

IKP	Parasitic knee current. Zero represents infinity.

The transforms are:

s2h	The function is TwoPort. The transform converts the de-embedded S-parameters to H-parameters.
H_calc	The function is VBIC_ac_solver. The transform solves for the H-parameters, given the four terminal voltages.
deembed_S	This is a program that de-embeds the S-parameters from the setup. It subtracts the dummy structure data measured in the delay Ypads setup: that is, it removes the parasitic capacitance of the pads. The output of this transform should be used for all subsequent extractions and optimizations in this setup.
opt_TR_QCO	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

real(H_calc.21)	real(s2h.m.21)
imag(H_calc.21)	imag(s2h.m.21)

	The optimization is performed over the full measurement range
opt_IKP	Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

real(H_calc.11)	real(s2h.m.11)
imag(H_calc.11)	imag(s2h.m.11)

The optimization is performed over the full measurement range

opt_TR_QCO_IKP Optimization. This optimizes the following simulated values to obtain the corresponding listed final values:

real(H_calc.11)	real(s2h.m.11)
imag(H_calc.11)	imag(s2h.m.11)
real(H_calc.21)	real(s2h.m.21)
imag(H_calc.21)	imag(s2h.m.21)

The optimization is performed over the full measurement range.
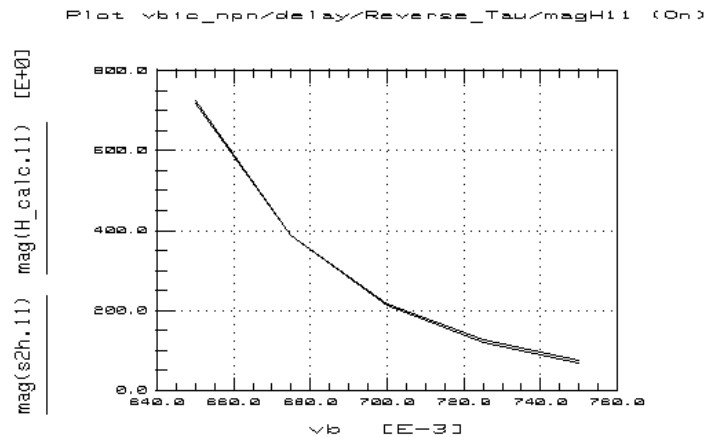
Figure 180   Example Plotted Reverse_Tau magH11 Data
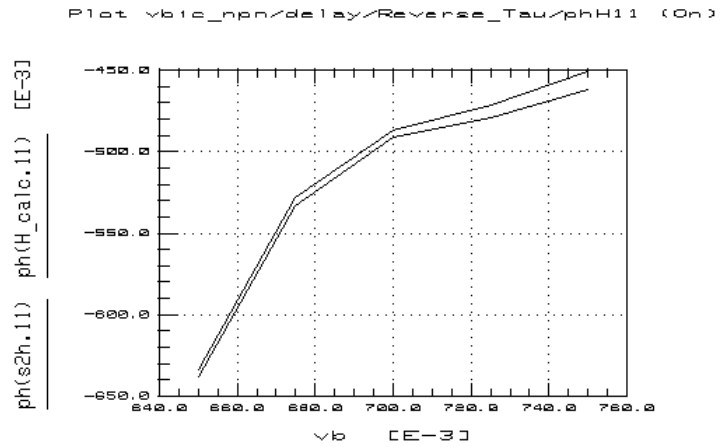
Figure 181 Example Plotted Reverse_Tau PhH11 Data
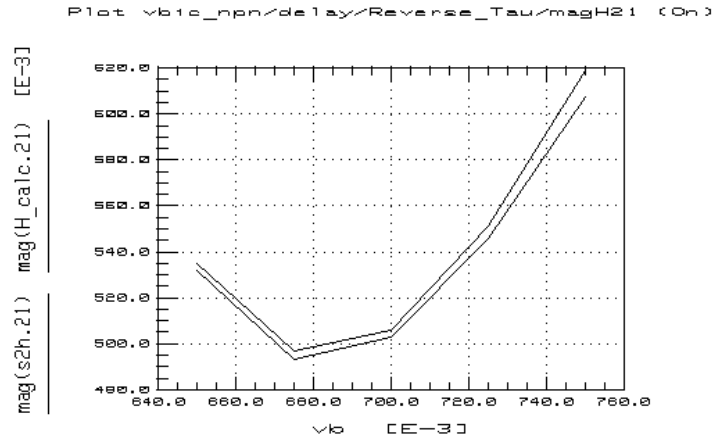
Figure 182 Example Plotted Reverse_Tau magH21 Data
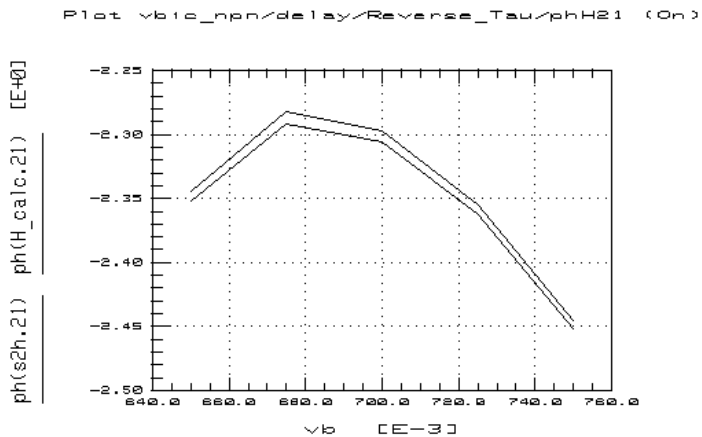
Figure 183 Example Plotted Reverse_Tau phH21 Data

Transforms in the delay Ypads Setup

No parameters are extracted. The measured data is used to de-embed the device S-parameters.

The transforms are:

s2y	The function is TwoPort. The transform converts the measured S-parameters to Y-parameters.
c11	This is a program that takes the output from the transform s2y and extracts the input capacitance value based on an equivalent pi structure for the parasitic capacitance.
c12	This is a program that takes the output from the transform s2y and extracts the crossover capacitance value based on an equivalent pi structure for the parasitic capacitance.
c22	This is a program that takes the output from the transform s2y and extracts the output capacitance value based on an equivalent pi structure for the parasitic capacitance.

Storing the Model

This procedure stores the model in IC-CAP, including all the device-specific measured and simulated data. Once you have developed the device model, it can be used as a component in the Agilent Microwave and RF Design System (MDS). You can use it for circuit simulation, inserting it into circuits under design to emulate the characteristics of your device.

If you are modeling large numbers of devices, it is worthwhile to organize your directory structure according to device type, geometry, and batch or wafer.

1 Use the UNIX directory you created at the beginning of the procedure, when you first opened the *vbic_npn.mdl* file.

2 From the Main window select **File > Save As**.

3 A dialog box is displayed, similar to the one illustrated.
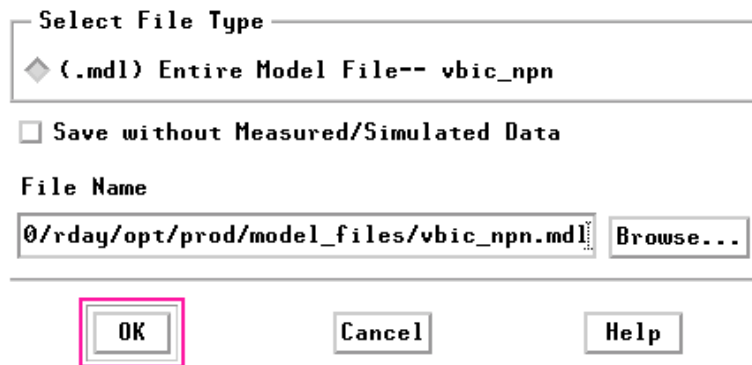
Figure 184   Save As Dialog Box

4 Select the File Type (.mdl for a complete model) by clicking the corresponding radio button.

5 Type in your chosen directory path and filename. Select **OK** to store the data and close the dialog box.

References

1 *VBIC95: An Improved Vertical, IC Bipolar Transistor Model.* Colin McAndrew, AT&T/Motorola; Jerry Seitchik, Texas Instruments; Derek Bowers, Analog Devices; Mark Dunn, Hewlett-Packard; Mark Foisy, Motorola; IanGetreu, Analogy; Marc McSwain, MetaSoftware; Shahriar Moinian, AT&T Bell Laboratories; James Parker, National Semiconductor; Paul van Wijnen, Intel/Philips; Larry Wagner, IBM.

2 *Avalanche Multiplication in a Compact Bipolar Transistor Model for Circuit Simulation.* W.J. Kloosterman and H.C de Graaff. IEEE 1988 BCTM.

3 *Spice Early Model.* McAndrew and Nagel. IEEE 1994 BCTM.

4 *A Survey of DC Methods for Determining the Series Resistance of Bipolar Transistors Including the New Delta ISub Method.* Jorg Berkner, SMI System Microelectronic Innovation GmbH, Frankfurt/Oder, Germany.

9 VBIC Model

A
Agilent 85190A IC-CAP

This appendix contains information on the software framework on which the high-frequency models are run.

The last part of this appendix is a procedure for getting started using the IC-CAP software, including launching IC-CAP and accessing the model file you require.

Agilent Technologies

The IC-CAP Framework

Agilent 85190A IC-CAP is the framework on which the model software runs. It provides the user interface, measurement system control, and simulator interface for the individual software models. You can find more information on the standard user interface and IC-CAP windows in the *User's Guide*.

Making Selections with the Mouse

This manual does not give instructions on how to use the mouse in the step-by-step procedures, therefore a summary is provided here.

When the terms *select*, *choose* and *click* are used, it means position the mouse pointer over the on-screen item and click the left mouse button. Sometimes you will be asked to select an item from a menu and then another item from a pulldown menu. In this case, do not release the mouse button after the first selection. Hold down the mouse button and move the pointer to the required selection in the pulldown menu, then release. A series of mouse selections separated by greater than symbols (>) indicates that the selections should be performed in sequence. An example is **File > Save As**.

If you need to get a readout of values at a particular point on the data trace, put the mouse pointer at this point and click middle to display the x- and y-axis values.

To edit information (such as variable names or values) in software, move the mouse pointer to the information to be edited. Click left with the mouse and move the pointer over the old information to highlight it. Then type in the new information over the old, and press Return. Use the Insert and Delete keys if necessary.

Software Backup

It is important that you make periodic backups of your system disk to guard against accidental loss of data. You can use the script */etc/backup* or the tar command to make backups. It is highly recommended that you make a tape backup of the system disk *before* you begin using the system.

System Shutdown

It is best never to switch your system off without doing a shutdown. UNIX systems have a very specific shutdown procedure that must be followed prior to switching the power off. Failure to follow the shutdown procedure can result in irreparable damage to the data on your disks. Refer to the documentation for your operating system, or obtain the shutdown procedure from your system administrator.

Launching IC-CAP

This assumes that IC-CAP has been properly installed on the computer. If you purchased the software together with the Hewlett-Packard workstation on which it will run, the software will have been installed at the Agilent factory. Otherwise, you can follow the installation procedures provided in the *Installation Guide*. Then proceed as follows:

1 Start IC-CAP from any terminal window by typing `iccap` and pressing Enter. The IC-CAP Main window appears (see the following figure).
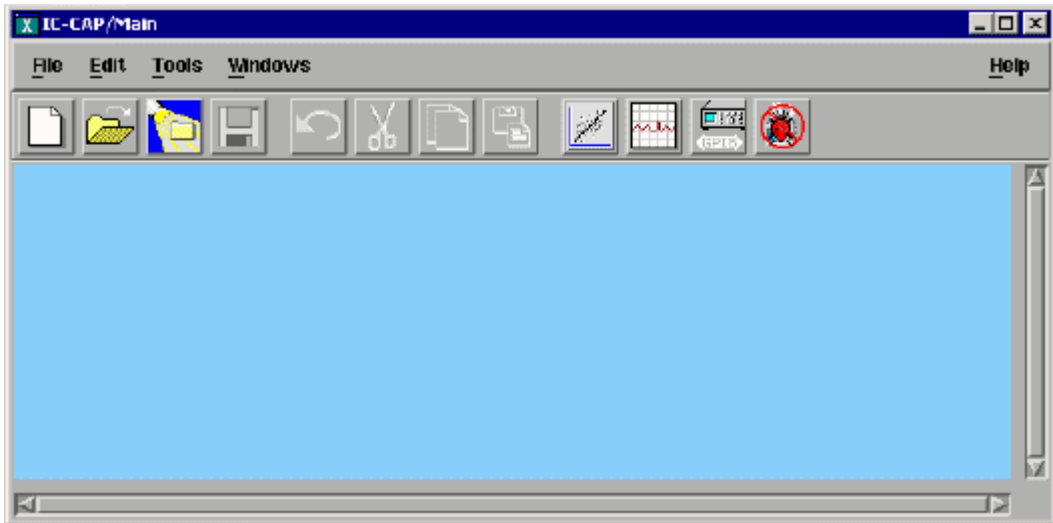
Figure 185   IC-CAP Main window

2 Select **File**, then select **Examples** from the pulldown menu. A list of model file types is displayed.

3 Double-click on the model file type you want, for example */bjt*. A list of model files of that type is displayed.

4 Scroll through the list to find the *.mdl* file for the model you wish to use, and double-click to open it or select it and click **OK**.

5 The Model window for the model of your choice is displayed.

6 Before you continue with the IC-CAP procedures, make a new directory in an appropriate location in the UNIX structure, using the mkdir command, to store the data and model files that will be generated. Give the directory a name that will associate it with the devices you are going to model.

7 Use the cd command to change to the new directory in UNIX.

8 To leave IC-CAP, choose **File > Exit**.

NOTE   The first time you load the model, warnings may be displayed in the UNIX window that the instrument options are stored. This merely indicates that the instrument state settings are being returned to default values. Once you set the appropriate instrument states for your measurements and save them in a file using the procedures in Chapter B, "Using IC-CAP with an Agilent 85122A Precision Modeling System or Chapter C, "Using IC-CAP with an Agilent 85123A Device Modeling System, you will not need to reset them each time you run the procedures.

Accessing Data from a Previous Measurement

If you wish to access data from a previously stored measurement or parameter extraction, you will need to perform this additional step to let IC-CAP know where to find the data and bring it into this model file:

From the IC-CAP Main menu, select **File > Change Directory**. Type in the pathname of the file where your data is stored. Click **OK**.

A   Agilent 85190A IC-CAP

B

Using IC-CAP with an Agilent 85122A Precision Modeling System

This appendix documents procedures for setup, calibration, and control of the Agilent 85122A precision modeling system. Some of these are hardware procedures independent of IC-CAP, and some are hardware control procedures performed inside IC-CAP.

Agilent Technologies

These procedures are common for the various high-frequency IC-CAP models. This is not a sequential set of procedures, but a set of separate hardware-related procedures that are used in conjunction with any of the device modeling procedures documented in the chapters. The procedures in this appendix give detailed instructions for the following:

- Configuring the system hardware
- Performing hardware setup in IC-CAP
- Calibrating the network analyzer
- Defining the instrument states in IC-CAP (for DC, capacitance, and S-parameter measurements)

In some cases, you may not need to perform all these procedures each time you model a device. The system setup procedures are performed rarely. On the other hand, the network analyzer calibration must be performed regularly. Setting the instrument states is done for each measurement setup in each model, and the settings can be stored.

Description of the System

The Agilent 85122A microwave parameter extraction system is specially designed for use with Agilent 85190-series high-frequency IC-CAP software (and a compatible controller) to measure the DC and high-frequency performance of active devices. The software is then used to extract the device model parameters. The standard system uses an Agilent 8510C network analyzer subsystem for S-parameter device characterization from 45 MHz to 20 GHz, and an Agilent 4142B DC source/monitor to provide precision DC characterization as well as bias for the S-parameter measurements.

The synthesized sweeper in the Agilent 8510 subsystem supplies RF signals. The Agilent 4142B DC source/monitor provides DC force (supply) and sense (measure) capability from its plug-in SMUs (source/monitor units). The DC signals are routed via cable feedthrough panels to the Agilent 11612 option Kxx external bias networks. The RF signals are also connected to the bias networks, and thus RF and DC signals are applied together to the device under test. The bias networks have 3.5 mm (female) connectors for interface to a test fixture or probe station.

The system can also be custom-configured to meet individual needs, to provide different operating frequency ranges, bias power levels, or cabling configurations, for example. Or it may include instrumentation for different types of measurements, such as power or noise figure measurements.

The system is factory-installed in a rack. A rack-mounted work surface is included, for maximum convenience in making on-wafer, in-fixture, or coaxial measurements. The work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.

The following figure shows the general configuration of a typical high-frequency modeling system, although the Agilent 85122A system is installed in a system rack.
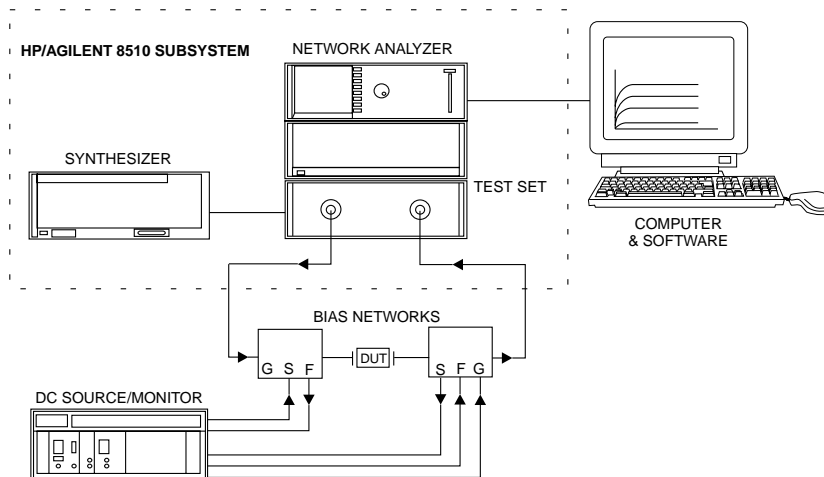
Figure 186   General Configuration of a Typical System

Configuring the System Hardware

The Agilent 85122A system is shipped from the factory with all rack components and instruments assembled and cabled in the system rack cabinet, and installed at your site by the Agilent customer engineer.

System hardware configuration need only be done in the following circumstances:

- At initial system setup.
- If changes are made to the system hardware.
- To change the bias connections for a different device type.

Before starting a parameter extraction procedure, make sure the GPIB and Agilent 8510 system cables are connected as shown in Figure 187, with the GPIB addresses set correctly. Note that the synthesizer and the DC source/monitor both use address 19: this is not a conflict, because the synthesizer is on the Agilent 8510 system bus and the DC source/monitor is on the GPIB. Make sure the synthesizer language is set correctly: the first three switch positions from the left on the synthesizer's GPIB/language switch must be set to 001 for network analyzer language.

Make sure the system RF cables are connected as shown in Figure 188.

If you are using a special custom system, refer to its installation and user's guide for custom cabling diagrams. The standard and special option Agilent 85122A installation and user's guides also provide any additional information you may need about setting up your system components.

Connect the RF cables from the test set ports to the RF IN ports of the bias networks.

The following provides details of the DC bias connections.
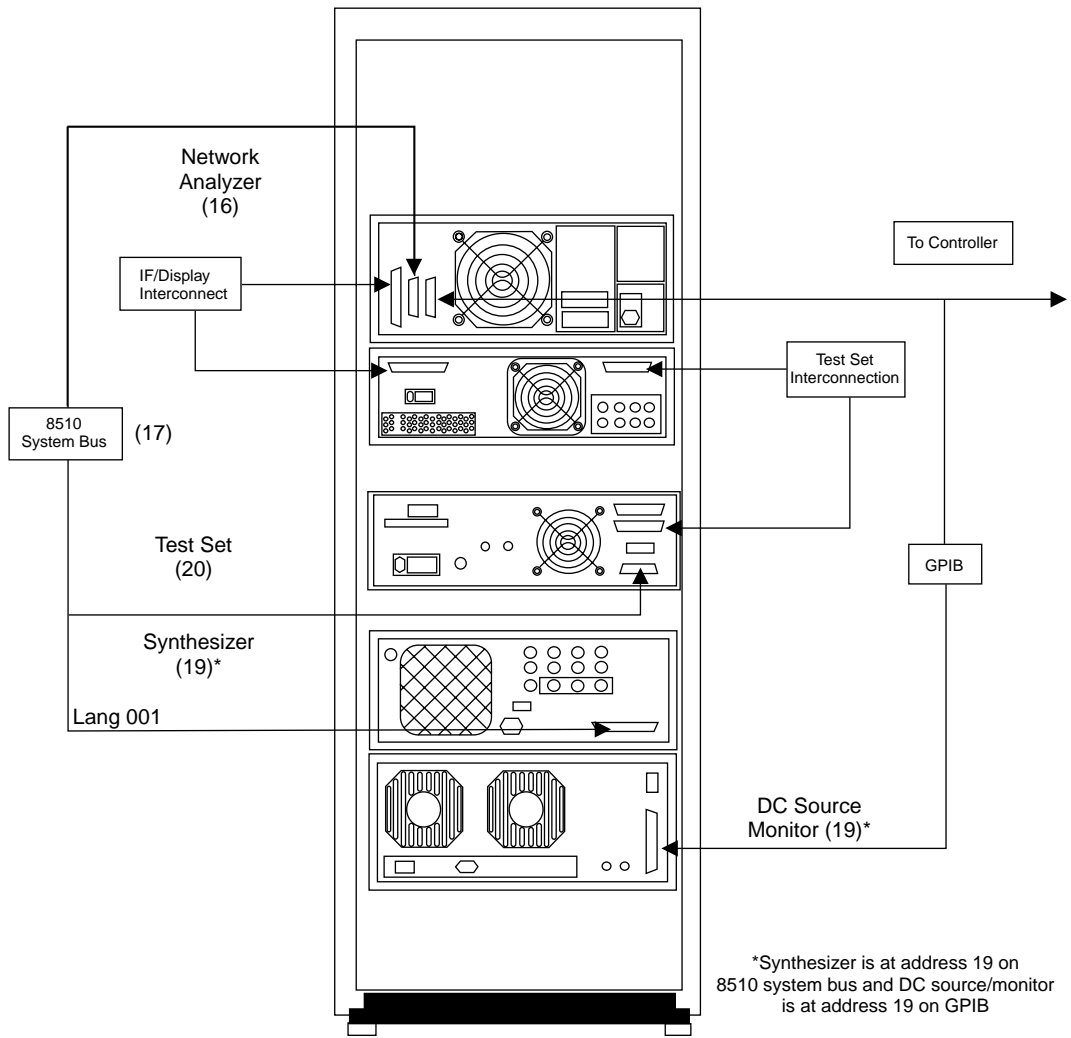
Figure 187 GPIB and System Interconnections and GPIB Addresses
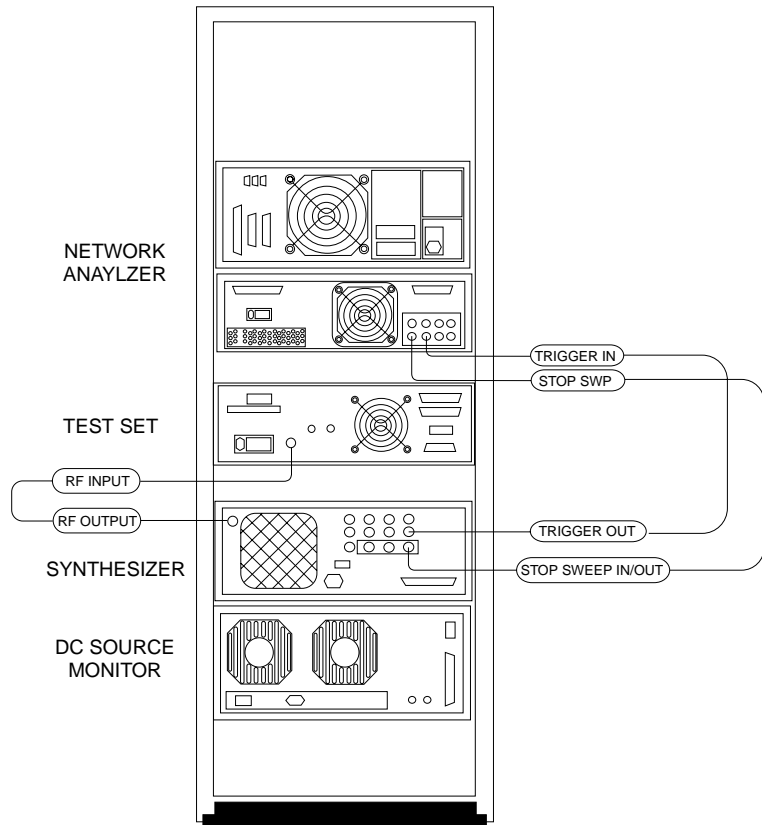
Figure 188   RF Cabling Interconnections

- For additional hardware information, refer also to the installation and user's guide for your Agilent 85122A microwave parameter extraction test system, and to the manuals for the individual instruments in the system.

- For information on fixtures or probe stations, refer to the manufacturer's documentation.

DC Bias Safety Considerations

Bias current and voltage are supplied to a device under test from the plug-in source/monitor units (SMUs) in the Agilent 4142B DC source/monitor, connected to the device through the bias networks and probes or fixture.

Interlock Connection

To prevent electric shock from DC voltages in excess of +/−42V, do not close the INTLK (interlock) connection of the Agilent 4142B DC source/monitor. The high-power SMU output can be as high as +/−200V, and the medium-power SMU output can be as high as +/−100V. As long as the INTLK connection is open, the DC voltage is clamped at +/−42 VDC.

Floating Ground

IC-CAP measurements are normally performed with the device in a floating-ground configuration, to prevent ground loop noise or, in the case of a BJT, possible damage to a device. A floating-ground configuration is accomplished by insulating the fixture or probe station from power-line ground, for example with an insulator between the wafer under test and the chuck of a probe station. If you are measuring in a floating-ground configuration, make sure the shorting bar of the Agilent 4142B is connected between the CIRCUIT COMMON and CHASSIS GROUND terminals on the front of the GNDU plug-in.

If you cannot implement a floating-ground configuration, it may be necessary to open the Agilent 4142B shorting bar and connect the DUT ground to the circuit common grounds at the DUT ends of the SMU and GNDU cables. The circuit common grounds are not connected through the bias networks: they are available at the input jacks. Refer to the Agilent 4142B *Operation Manual* chapter 3, "Test Device Connections," and read the section titled "If the Test Device Is Externally Grounded (Floating Measurement)." This provides detailed instructions on making the necessary connections.

WARNING   A potential shock hazard exists when the shorting bar is disconnected for floating measurements. Do not touch any of the Agilent 4142B front panel connectors while a floating-ground measurement is in progress.

Configuring the SMUs and Bias Networks

Bias current and voltage are supplied to a device under test from plug-in source/monitor units (SMUs) in the Agilent 4142 DC source/monitor, connected to the device through the bias networks and probes or fixture. The standard Agilent 85122 system includes one medium-power SMU and one high-power SMU.

The quadraxial cables from the SMUs are routed through a cable feedthrough panel in the system rack, around the back of the synthesizer and test set, and out to the front of the rack through another feedthrough panel.

Bias Connections

The following figure illustrates the connections from the SMUs to the bias networks. Connect the FORCE and SENSE connectors on the quadraxial cables to the FORCE and SENSE connectors on the bias networks. Connect the triaxial cable from the GNDU SMU to the ground connector on one of the bias networks (usually port 2). Leave the ground connector of the other bias network unconnected.
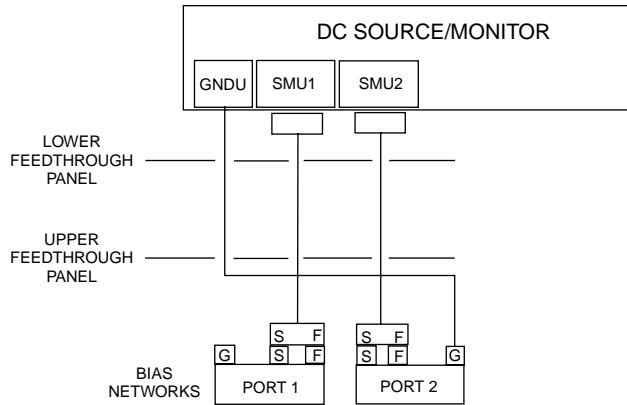
Figure 189   Connections from DC Source/Monitor to Bias Networks

Connections from Bias Networks to Device

> **NOTE**   For in-fixture measurements, the measurement procedures in this manual apply to a FET mounted in the fixture in a common-source configuration, or a BJT mounted in a common-emitter configuration.

Connect the DC/RF signals from the outputs of the bias networks via the semi-rigid cables to the corresponding terminals of the fixture or probe station. Set up the angle of the cables and bias modules to minimize stress and torque, and ensure that the cables are properly supported. Iterate the positioning of the cables and bias networks until the connectors line up correctly. Use a torque wrench to tighten the connections.

The following figure illustrates the connections for a BJT. The connections for a FET are comparable, with the port 1 DC/RF signal connected to the gate, the port 2 DC/RF signal connected to the drain, and the source grounded.
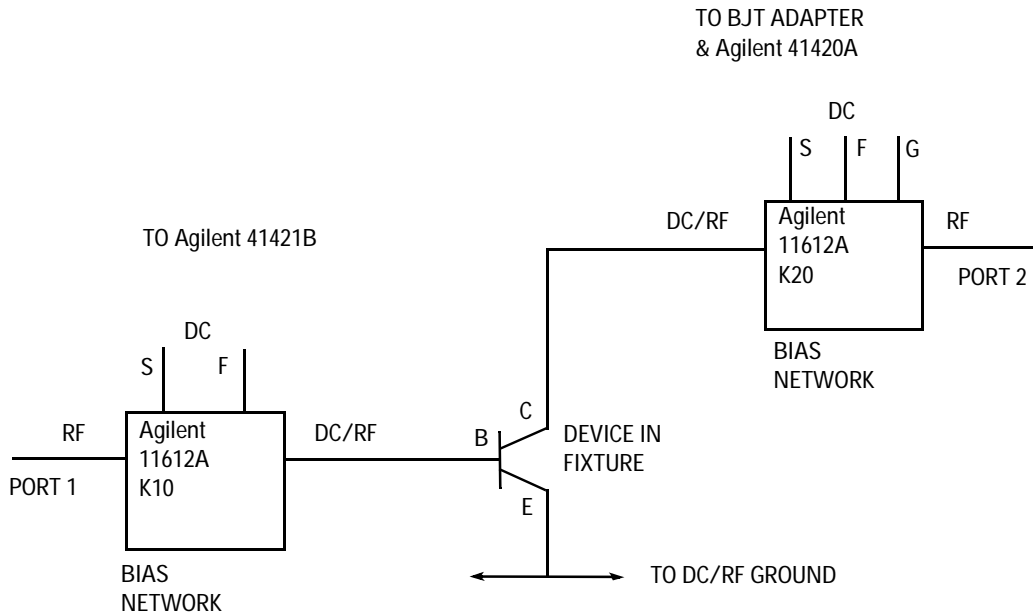
Figure 190   Connections from Bias Networks to In-Fixture DUT

Switching on the System

It is important to apply power to the system in the correct sequence, so that the network analyzer recognizes the test set and synthesizer, and the computer recognizes the network analyzer subsystem and the DC source/monitor.

Switch on power to the system instruments in this sequence (from bottom to top of the system rack):

1 DC source/monitor

2 Synthesizer

3 Test set

4 Network analyzer

5 Computer

Preparing for Calibration

Let the system warm up for at least 1 hour before calibrating the network analyzer. The DC measurements can be performed during this time.

The calibration standards will need to be at ambient room temperature when the measurement calibration is performed. If you are using a fixture, open the fixture calibration kit and place all the devices on top of the foam so they will reach room temperature by the time the system is warmed up. If you are using a probe station, place the probed impedance standard substrate (ISS) on the probe chuck and allow it to reach ambient temperature before use.

The calibration procedure is described in "Calibrating the Network Analyzer" on page 589.

Terminating the Input Ports to Prevent Oscillation

Leave the network analyzer disconnected from the bias networks while you perform the DC measurements. Connect terminations with >20 dB return loss (such as 10 dB pads) to the RF IN connectors of the bias networks, to prevent possible bias oscillation. The network analyzer is calibrated later in the modeling procedures, immediately before the capacitance and S-parameter measurements.

Alternatively, isolate the device from RF power by manually setting 90 dB of attenuation at the network analyzer test ports. Also put the network analyzer in hold sweep mode by pressing STIMULUS > MENU > MORE > HOLD.

Installing the Device

The best time to install the device is immediately before making measurements.

CAUTION	Ground yourself with an antistatic wrist strap to reduce the chance of electrostatic discharge. A ground terminal is provided on the lower left corner of the test set front panel for your convenience. (Note that the system work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.)

Install the device carefully on the probe chuck or in the fixture. Be sure to handle the device as little as possible, to avoid damaging it. Once the device is in place, avoid bumping the test station.

NOTE	Eliminate direct light sources (such as microscope light) during measurement, especially for GaAs devices, as light may influence the measurement and cause inaccurate results.

Performing Hardware Setup in IC-CAP

The measurement system hardware is controlled by the IC-CAP software. This procedure configures IC-CAP to recognize the system instruments on the GPIB, and the individual source/monitor units (SMUs) in the DC source/monitor. The complete procedure need only be done after initial system installation, or any time the system hardware is changed, to establish a system handshake. The second part of the procedure (renaming the SMUs) also needs to be done any time you change device types, such as from FET to BJT or from BJT to FET.

For help with launching IC-CAP and opening the model file you need, refer to Chapter A, "Agilent 85190A IC-CAP," then continue with these steps:

1 In the IC-CAP Main menu bar select **Tools > Hardware Setup**. The hardware window is displayed. It will vary depending on the system hardware used, but will be similar to the following figure.
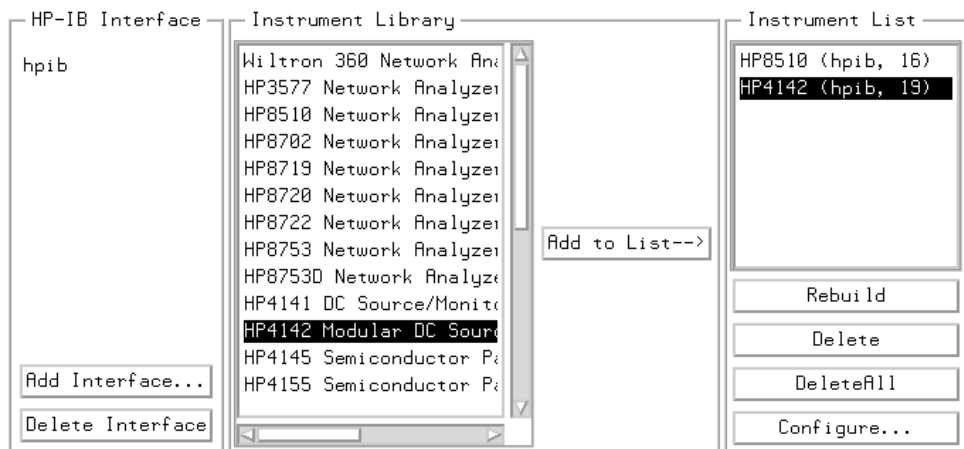
Figure 191   Hardware Window

2 Under the Instrument List select **Rebuild**. This command causes IC-CAP to poll all available GPIB addresses and add to the Instrument List any newly connected instruments that

are powered up. (It does not affect instruments already on the list, whether they are powered up or not.)

Renaming the SMUs

Use this procedure any time you change device types, such as from FET to BJT or from BJT to FET, so that the measurement setups can properly communicate with the bias supplies.

When you run the IC-CAP software, it initially identifies the plug-in SMUs according to the numbers of the Agilent 4142 slots in which they are installed. A medium-power SMU occupies one slot. A high-power SMU occupies two slots, and is identified by the higher slot number of the two. An Agilent 85122 system can include different combinations of SMUs: the standard system used in the example procedures includes two SMUs, one Agilent 41421B medium-power SMU and one Agilent 41420A high-power SMU. The medium-power SMU is factory-installed in slot 1 and is initially identified in the IC-CAP software as MPSMU1. The high-power SMU is installed in slots 2 and 3, initially identified as HPSMU3. The ground unit is identified as GNDU. It is generally more convenient to assign unit names to the SMUs that identify their purpose in a device measurement, depending on the bias connections at the device terminals and the type of device you are modeling.

In configuring IC-CAP to recognize the system hardware, you need to set the SMU names used in software to identify the SMU connections at the measurement terminals. In a FET measurement, the SMUs are identified as VG (gate supply) and VD (drain supply). In a BJT measurement, the SMUs are identified as SMU1 and SMU2, corresponding to the network analyzer test ports: this helps to facilitate measurement of a BJT with only two SMUs, since the port 2 SMU will be used for measurements at both the collector and emitter terminals.

The following figure illustrates the SMU unit names for a FET measurement and the correspondence to the device terminal connections.
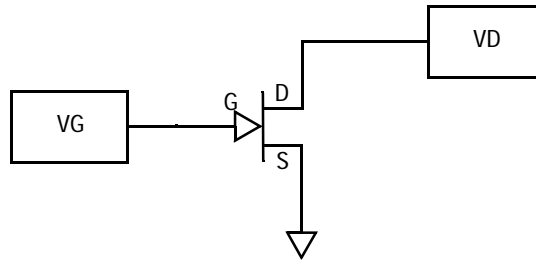
Figure 192   SMU Unit Names for a FET Measurement

The following figure illustrates the SMU unit names for a BJT measurement and the correspondence to the device terminal connections
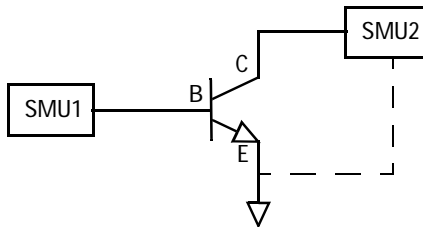
Figure 193   SMU Unit Names for a BJT Measurement

1 In the Instrument List click on **HP4142** to highlight it. Select **Configure** and the Configuration of HP4142 window is displayed.

2 The Unit Table lists the SMUs. Also listed for each is a unit ID (identification) name that can be edited.

In a FET measurement, the medium-power MPSMU1 is renamed VG, because it is connected to the gate terminal; and the high-power HPSMU3 is renamed VD, because it is connected to the drain terminal.

In a BJT measurement, the medium-power MPSMU1 is renamed SMU1 because it is the port 1 SMU; and the high-power HPSMU3 is renamed SMU2 because it is the port 2 SMU.

3 To make this change, click left with the mouse and move the mouse pointer over the name assigned to MPSMU1 to highlight it. Then type in the new name, and press Return. Similarly, change MPSMU3's assigned unit name. Then select **OK** to close the configuration window.

4 You can close the Unit Table tile by clicking middle again.

5 To save this hardware configuration in a file for future use, select **Hardware**, then **Write to File**. A file filter is displayed, showing the pathname of the current directory. Type in an appropriate name for the *.hdw* file you are creating, and select **OK**.

6 Close the hardware window.

After IC-CAP is configured to recognize the system hardware, you can proceed to one of the modeling chapters.

Calibrating the Network Analyzer

The network analyzer must be calibrated before any S-parameter measurements are performed. The system must be allowed to warm up for at least 1 hour before calibration, and the calibration standards must be at ambient room temperature. It is convenient to switch on the analyzer and expose the calibration devices to air before you make the DC measurements.

Good calibration of the network analyzer system is critical to a good high-frequency measurement and extraction. A good calibration is dependent on the quality of the calibration kit standard devices, the care with which they are maintained, and the correctness and repeatability of the device connections. A stable ambient temperature ($\pm 1°C$) is required to maintain a good calibration.

For the high-frequency IC-CAP measurement procedures, the network analyzer is calibrated over a broadband frequency range, to perform the S-parameter measurements for parasitic and AC extractions. In addition, one or two subset calibrations may be needed at CW frequencies for specific measurement setups. This procedure explains how to set up both the broadband calibration and one of the CW calibrations with one set of standards measurements, making the CW frequency cal a subset of the broadband frequency cal. If a third calibration is done, its frequency may not be known until some of the S-parameter measurements have been made, therefore it cannot be performed now. However, when the calibration frequency is determined later, the subset cal can be performed using exactly the method defined under "CW Frequency Calibration Subset" on page 591 without another set of standards measurements.

The instructions given below are very abbreviated: more detailed information is available in the Agilent 8510 *Operating and Programming Manual*.

NOTE

It is important that values entered in software correspond with actual instrument settings. Be sure to write down the parameters of your calibrations, so that they are readily available when you need to enter them in IC-CAP.

To connect the network analyzer:

Disconnect the terminations from the bias networks. Connect the RF cables from the network analyzer test ports to the RF inputs of the bias networks.

Swept Broadband Calibration

1 On the network analyzer, press PRESET.

2 To define the frequency range of the calibration and measurement, on the network analyzer press STIMULUS > START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as G/n) at the right of the keypad. Similarly, press STOP and set the stop frequency. Set a frequency range at least as wide as the operating range of the device.

3 Press STIMULUS > MENU > NUMBER of POINTS, and set the number of data points you wish to measure. A reasonable number might be 51 or 101 points.

4 Press PRIOR MENU > FREQUENCY > STEP to set the stepped frequency mode, which provides greatest measurement accuracy by phase-locking the synthesized sweeper at each frequency point.

5 Press PRIOR MENU > MORE > CONTINUAL to set a continual stimulus sweep.

6 Press PRIOR MENU > POWER MENU, and set the desired power level. In setting the source power and attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If the power level at the sampler goes above −8 dBm, an IF OVERLOAD error message is displayed and

you will need to reduce the source output power. The default network analyzer power level is 0 dBm.

7 For a device with power dropoff at higher frequencies, you may wish to set a power slope using the stimulus menus. An appropriate power slope would be in the region of 2-3 dB/GHz.

8 Press RESPONSE > MENU > AVERAGING ON/restart, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. The default averaging factor is 256, but as little as 16 may be adequate.

9 If the cal kit constants for your calibration kit are not loaded into the network analyzer, load them from disk now.

10 If you wish to modify one of the internal calibration kit definitions (see below), do so now.

11 Perform a full two-port calibration using short-open-load-thru standards. Omit isolation cal. (A TRL or LRM calibration would also be an appropriate alternative.)

12 At the end of the calibration sequence, the cal set numbers are listed in the softkey menu. Press CAL SET #1 to store this calibration in cal set #1. (Or use another available cal set.)

CW Frequency Calibration Subset

In some of the high-frequency models, extraction of certain parameters is done at a single CW frequency. The CW frequency must be equal to the start frequency plus an integer multiple of the step size: the model chapter you use gives more information on deciding an appropriate frequency. Once you have determined the frequency of the calibrated measurement point, use the following procedure to create a CW frequency subset of the swept broadband calibration.

1 Press CAL > CORRECTION ON > CAL SET 1 (this turns on the swept cal you just completed), then MORE > MODIFY CAL SET > FREQUENCY SUBSET.

2 Use the SUBSET: START and SUBSET: STOP softkeys in the SUBSET menu (NOT the front panel keys) to set both start and stop to the frequency you selected.

3 Press CREATE & SAVE, and store this calibration in cal set #2 (or another available cal set).

After the network analyzer is calibrated, return to the modeling chapter you are using, for the S-parameter measurement procedures.

Modifying a Cal Kit Definition

The network analyzer's internal calibration kit models are accurate for most applications, however, you may want to modify the internal cal kit definitions in certain situations, for example:

• If you are using a probe station and you know the probe capacitances, you can enter them into the cal kit definition so that their effects can be removed by the measurement calibration.

• The impedance of an on-wafer load may not be exactly 50Ω. You can characterize the DC impedance of a load exactly with a precision ohmmeter measurement, and enter the impedance value into the cal kit definition.

The procedure to modify an internal cal kit definition is complex, and not to be undertaken without a solid understanding of error correction and the system error model. It is explained in *Product Note 8510-5, Specifying Calibration Standards for the Agilent 8510 Network Analyzer* (Agilent literature number 5954-1559), and in the Agilent 8510C *Operating and Programming Manual*.

More on Power Levels and Capacitance Measurements

The primary factor to consider in understanding the available range of power is the S-parameter test set, and whether it is coupler-based or power-splitter-based. The test set generally used in a 26.5 GHz system is the Agilent 8515B, which uses a power splitter to couple signals to the detector. The "economy" version test set is the Agilent 8514B (20 GHz), which uses a directional coupler.

The directional coupler in the Agilent 8514B has a −20 dB/decade power rolloff below 500 MHz, with some signal loss above that. The result is that in the frequency range 45 MHz to 2 GHz the minimum detectable signal of the Agilent 8514B is only −66 dBm. However, over the same frequency range, the minimum detectable signal of the Agilent 8515B is −101 dBm. (Above 2 GHz, both test sets can detect at least −95 dBm signals.)

The model of synthesizer used does not affect these differences. Any potential improvement due to the synthesizer is obscured by the effects of the test set.

Capacitance Junction Measurements

In measuring capacitance junctions, the signal level at the device is generally kept below −30 dBm to keep nonlinearity-induced errors below 5 percent. Capacitance measurements are typically made in the 50 to 100 fF region.

Why is it necessary to keep the signal level below −30 dBm for a capacitance measurement? In fact, it is *not* necessary! The device is biased so as not to have any gain, therefore the input samplers and the test set are not saturated, nor is the output of the device. Additionally, the power level set on the synthesizer is not the same as the power level arriving at the device. Depending on the frequency range and the test set, there can be anywhere from 15 to 30 dB of loss between the source and the device.

One method is to solve for the minimum resolvable capacitance, given an input power and a dynamic range. This should be done for both S21 (Cbc) and S11 (Cbe), because their respective resolutions are different.

Device input power −10 dBm.

Noise Floor −90 dBm.

Dynamic range >80 dB. (80 dB = 1E−04 linear)

S21 Related to Cbc

$$|S21| \; = \; \frac{2 \cdot Z_0}{(2 \cdot Z_0 + 1 / \omega C)} \; = \; 1 \cdot 10^{-04}$$

$$C \; = \; \frac{|S21|}{(2 \cdot \omega \cdot Z_0 (1 - |S21|))}$$

= 1.5917E-07/freq

=> minimum C @ 100 MHz = 1.59 fF

S11 Related to Cbe

$$|S11| = \frac{(1/j\omega C) || (Z_0 - Z_0)}{(1/j\omega C) || (Z_0 + Z_0)} \; = \; 1 \cdot 10^{-04}$$

$$C \; = \; \frac{2|S11|}{Z_0 \cdot \omega (|S11| + 1)}$$

=6.36556E-07/freq

=> minimum C @ 100 MHz = 6.37 fF

This analysis verifies the observation that Cbc measurements are cleaner than Cbe measurements.

With an Agilent 8515B test set the same capacitance would put the signal 40 dB above the noise floor, which would allow 20 dB of leeway even as low as 100 MHz.

This verifies the initial statement that it is the S-parameter test set that determines the available range of power, and not the synthesized sweeper.

Defining the DC Source/Monitor Instrument State

NOTE For details on IC-CAP instrument options (instrument states) for specific instruments, refer to Chapter 1, "Supported Instruments," in the *Reference* manual.

IC-CAP provides remote control of the DC source/monitor and the network analyzer to measure the device under test. This procedure configures IC-CAP with the instrument state settings (instrument options) for the DC source/monitor.

Use this procedure if you have been referred here from one of the main device modeling procedures.

You will need to set the DC source/monitor instrument state separately for *each* measurement setup in a modeling procedure. However, use the same settings for all the measurement setups in one model (except that Integ Time can differ among setups, as explained below). Unless you use the default settings documented here, it is helpful to write down the settings to duplicate in the other setups.

NOTE DC signal levels and other input parameters that differ among measurement setups are not set in this procedure, but in the individual setup procedures in each model chapter.

This procedure uses the forward gummel setup in the BJT measurement as an example. However, the settings used are defaults and the explanations are general.

1 In the DUT/Setup panel select a setup, for example fgummel.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states for the DC source/monitor, as illustrated in the following figure.

3 Set each of the DC source/monitor measurement parameters according to the actual Agilent 4142 configuration and the device to be measured, using the guidelines in the following steps.

Figure 194 Example Agilent 4142 DC Source/Monitor Instrument State

- Use User Sweep is normally set to No, to use the instrument's internal sweep, which is faster than a user-defined sweep.

- Hold Time is the delay in seconds before the sweep, to allow for DC settling. Generally, no hold time is necessary.

- Delay Time is the time in seconds the instrument waits before measuring at each step of an internal or user sweep. It can generally be set to 0.000.

- For Integ Time, M (medium) is a good default choice. In measurements where a long integration time is needed for noise reduction, you would use L (long), and you will be instructed to do so in certain procedures. An example is the forward gummel measurement, which measures very low current values. You can speed up a measurement by using S (short), but this is not recommended because it degrades the dynamic range of the measurement.

4 Set Range to 0 to implement SMU auto-ranging.

5 Power Compliance is used to set the maximum current/voltage combination for the DC source/monitor. However, in the models, SMU voltage and current compliances are set individually in the individual setups. Therefore the value here can be set to 0.000.

6 SMU Filters ON is set to Yes to switch in low-pass filters on the SMU outputs. This protects the device from voltage spikes caused by DAC output changes.

7 The IC-CAP models documented here are generally not configured for pulsed measurements, therefore Pulse Unit can be left blank. All other Pulse settings are then irrelevant and can be ignored.

8 Module Control is not used in these procedures: leave the field blank.

9 Init Command sets the instrument to a mode not supported by other fields in this table. It is not used in these models. Leave the field blank.

10 If you wish, refer to the DC source/monitor manual for more detail.

11 If the measurement setup you are configuring calls for a DC measurement only, close the instrument options window. Then return to the modeling procedure.

12 If the measurement setup also calls for a network analyzer measurement, continue to "Defining the Instrument States for S-Parameter Measurements" on page 598.

Defining the Instrument States for S-Parameter Measurements

IC-CAP provides remote control of the DC source/monitor and the network analyzer to measure the device under test. These procedures configure IC-CAP with both the DC source/monitor and Agilent 8510 network analyzer instrument states for S-parameter measurements. The first procedure sets the instrument states for a broadband swept-frequency measurement, and the second for a CW frequency measurement.

> **NOTE**   For details on IC-CAP instrument options (instrument states) for specific instruments, refer to Chapter 1, "Supported Instruments," in the *Reference* manual.

Instrument States for Swept Measurements

You will need to set the instrument states separately for *each* measurement setup in a modeling procedure. However, use the same settings for all the swept S-parameter measurement setups in one model (except that Integ Time can differ among setups, as explained under "Defining the DC Source/Monitor Instrument State" on page 595) Unless you use the example settings documented here, write down the settings you use, to duplicate among setups within one model.

This procedure uses the sparm_bias setup in the BJT measurement as an example, though the explanations here apply to all swept-frequency S-parameter measurements in the different device measurement procedures.

> **NOTE**   Source parameters such as frequency range are not set in this procedure, but in the individual S-parameter setup procedures in each model chapter.

NOTE	Do not use these settings for a CW S-parameter measurement. The settings for a CW measurement are different, and are documented later.

1 In the DUT/Setup panel select the S-parameter setup you need, for example sparm_0v.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states. The following figure illustrates example instrument states.

Figure 195 Instrument States for Swept Measurements

3 Set the Agilent 4142 instrument state the same as for the DC measurements.

4 Set the Agilent 8510 instrument state according to the actual hardware configuration, the calibration used, and the device to be measured, following these guidelines:

5 Set Use User Sweep to No, to use the instrument's standard internal sweep. This is necessary for the network analyzer's calibration to be switched on.

6 Hold Time is the time in seconds before the instrument starts a sweep, to allow for DC settling. Generally, no hold time is required.

7 Delay Time is the delay in seconds the instrument waits before setting each frequency in user sweep mode. The default is 100.0m, but the value set will not affect an internal sweep.

8 Set Port 1 Atten and Port 2 Atten to the same levels in dB that you set in the network analyzer calibration. While the default is 0 dB, you may need to add attenuation for measurements of amplifiers or high-power devices.

9 Source Power is the RF synthesizer output power. Set it to the same value in dB that you set in the network analyzer calibration.

10 Power Slope is used only if you use power slope in the network analyzer calibration: if so, set the corresponding value here. This is useful to view the response of a device with power dropoff at higher frequencies. The units for power slope are dB/GHz; the default value is 0.000. (A network analyzer message will caution you that the correction may be invalid, but this can be ignored.)

11 Set Fast Sweep (RAMP) to No, because the network analyzer is in stepped-sweep mode, set earlier in the calibration procedure.

12 Sweep Time applies only to ramp sweep mode, therefore the value set is irrelevant for a stepped-sweep measurement.

13 Set Use Fast CW to Yes, to minimize repeated switching between the test set ports.

14 Trim Sweep is set to 0. This feature is used only in ramp sweep mode.

15 Set Avg Factor to the same averaging factor you set in the calibration. The default value is 256, but as little as 16 may be adequate.

16 Set Cal Type (SHN) to H, for hardware.

17 Set Cal Set No to the cal set number in the analyzer where you stored your swept calibration, so that IC-CAP can find the calibration.

18 Soft Cal Sequence refers to the sequence of measurements of the cal standard devices: load-open-short-thru.

19 Delay for Timeouts can generally be set to the default value of 0.000.

20 Set Use Linear List to No because this is a standard stepped-frequency measurement.

21 Init Command sets the instrument to a mode not supported by other fields in this table. It is not used in this model. Leave the field blank.

22 Close the instrument options window. Then return to the modeling procedure.

Instrument States for CW Frequency Measurements

The instrument states for a CW S-parameter measurement are similar to those for a swept measurement, with a few important exceptions, listed below.

You will need to set the instrument states separately for *each* measurement setup in a modeling procedure. However, use the same settings for all the CW S-parameter measurements in one model (except that Integ Time can differ among setups.) Unless you use the default settings documented here, write down the settings you use, to duplicate among setups within one model.

This procedure uses the sparm_cje setup in the BJT measurement as an example, though the explanations here apply to all CW S-parameter measurements in the different device measurement procedures.

NOTE	Frequencies are set in the setup Inputs, therefore they do not affect instrument states even if you choose to use different CW frequencies for different measurements, except for the Cal Set No where they are stored.

1 In the DUT-Setup panel select the CW S-parameter setup you need, for example sparm_cje.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states. Figure 196 illustrates example instrument states.

3 Set the Agilent 4142 instrument state the same as for the DC measurements.

4 Set Use User Sweep to Yes, because CW is a nonstandard sweep.

5 (Be sure to set Port 1 Atten and Port 2 Atten and Source Power to the same values as you used in the network analyzer calibration.)

6 Set Use Fast CW to No.

7 Set Cal Set No to the cal set number in the analyzer where you stored your CW subset calibration. This must be different from the cal set used by the swept broadband calibration, and from any CW calibration you make at a different frequency for any different measurement.

Figure 196   Instrument States for CW Frequency Measurements

8 If necessary, refer to the Agilent 8510C *Operating and Programming Manual* for more detail.

9 Close the instrument options window. Then return to the modeling procedure.

B   Using IC-CAP with an Agilent 85122A Precision Modeling System

C

Using IC-CAP with an Agilent 85123A Device Modeling System

This appendix documents procedures for setup, calibration, and control of the Agilent 85123A RF parameter extraction test system. Some of these are hardware procedures independent of IC-CAP, and some are hardware control procedures performed inside IC-CAP.

These procedures are common among the high-frequency IC-CAP models. This is not a sequential set of procedures, but a set of separate hardware-related procedures used in conjunction with any of the device modeling procedures documented in the modeling chapters. Use the information in this appendix when you are directed here, at the appropriate times in the modeling procedures.

Agilent Technologies

The procedures in this appendix give detailed instructions for the following:

- Configuring the system hardware
- Performing hardware setup in IC-CAP
- Defining the instrument states in IC-CAP (for DC, capacitance, and S-parameter measurements)
- Calibrating the network analyzer

The instructions explain when the hardware-related procedures are needed and when they are not. In some cases, you may not need to perform all these procedures each time you model a device. The system setup procedures are performed rarely. Setting the instrument states is done for each measurement setup in each model, and the settings can be stored for repeated use. The network analyzer calibration must be performed regularly for accuracy.

NOTE	The instructions provided in this appendix are for an Agilent 8753 network analyzer. The procedures for an Agilent 8720 are very similar. For information specific to a different network analyzer, refer to its operating manual.

Description of the System

The Agilent 85123A RF parameter extraction test system is specially designed for use with Agilent 85190-series high-frequency IC-CAP software (and a compatible controller) to measure the DC and RF performance of active devices. The software is then used to extract the device model parameters. The standard system uses an Agilent 8753D network analyzer subsystem for S-parameter device characterization, and an Agilent 4142B DC source/monitor to provide precision DC characterization as well as bias for the S-parameter measurements. The Agilent 8753D subsystem consists of the network analyzer with its integrated synthesizer and S-parameter test set, the RF cables, and the calibration kit and verification kit if used.

The integrated synthesizer in a standard Agilent 8753D network analyzer supplies a swept or CW RF signal in the range of 30 kHz to 3 GHz. The integrated test set separates the RF signal into reference and test signals, and provides RF connections via cables and adapters to the external bias networks. The Agilent 4142B source/monitor provides DC force (supply) and sense (measure) capability from its plug-in SMUs (source/monitor units). The DC signals are routed via feedthrough panels to the bias networks, and thus RF and DC signals are applied together to the device under test. The bias networks have 3.5 mm (female) connectors for interface to a test fixture or probe station. The transmitted and reflected signals from the device are measured and displayed by the receiver.

The system can be custom-configured to meet individual needs, for example to provide an optional 6 GHz frequency range, different bias arrangements, or different cabling configurations. The network analyzer can be replaced with an Agilent 8720 (20 GHz) or 8719 (13.5 GHz). Or the system can include instrumentation for different types of measurements, such as power or noise figure measurements.

The system is factory-installed in a rack. A rack-mounted work surface is included, for maximum flexibility and convenience in making on-wafer, in-fixture, or coaxial measurements. The work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.

Configuring the System Hardware

The Agilent 85123A system is shipped from the factory with all rack components and instruments assembled and cabled in the system rack cabinet.

System hardware configuration need be done only in the following circumstances:

- At initial system setup.
- If changes are made to the system hardware.
- To change the bias connections for a different device type.

Before starting a parameter extraction procedure, make sure the GPIB cables are connected as shown on the next page, with the GPIB addresses set correctly.

If you are using a special custom system, refer to its installation and user's guide for custom cabling diagrams. The standard and special option Agilent 85123A installation and user's guides also provide additional information you may need about setting up your system components.

Connect the RF cables from the test ports to the RF IN ports of the bias networks, using adapters as needed.

The following provides details of the DC bias connections.
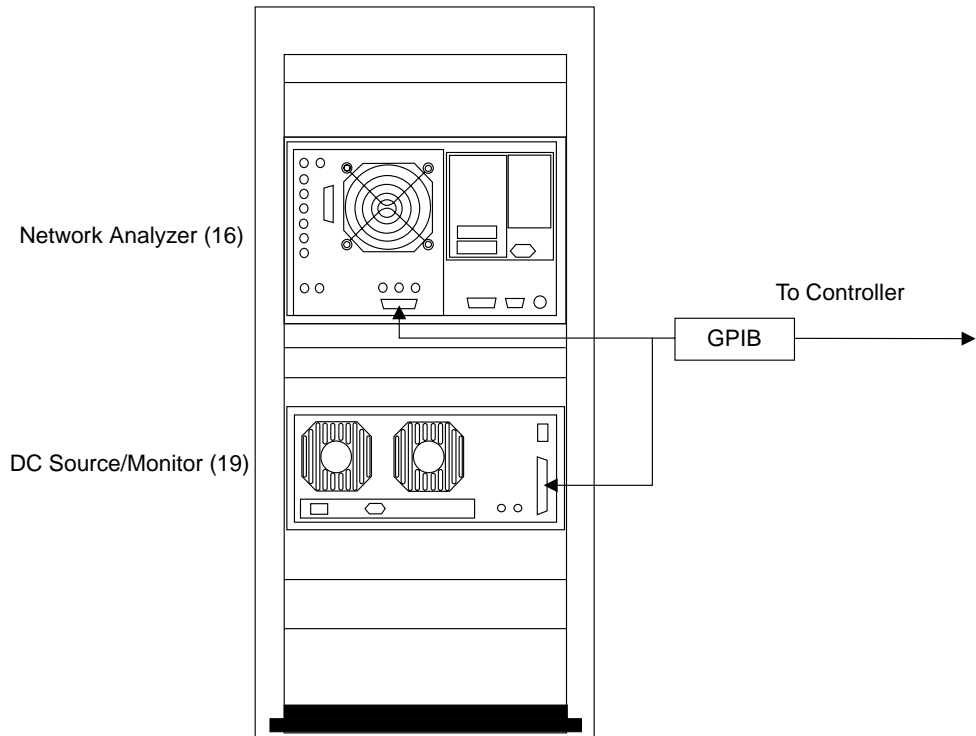
Figure 197   GPIB Interconnections and GPIB Addresses

DC Bias Safety Considerations

Bias current and voltage are supplied to a device under test from the plug-in source/monitor units (SMUs) in the Agilent 4142B DC source/monitor, connected to the device through the bias networks and probes or fixture.

Interlock Connection

To prevent electric shock from DC voltages in excess of +/−42V, do not close the INTLK (interlock) connection of the Agilent 4142B DC source/monitor. The high-power SMU output can be as high as +/−200V, and the medium-power SMU output can be as high as +/−100V. As long as the INTLK connection is open, the DC voltage is clamped at +/−42 VDC.

Floating Ground

IC-CAP measurements are normally performed with the device in a floating-ground configuration, to prevent ground loop noise or, in the case of a BJT, possible damage to a device. A floating-ground configuration is accomplished by insulating the fixture or probe station from power-line ground, for example with an insulator between the wafer under test and the chuck of a probe station. If you are measuring in a floating-ground configuration, make sure the shorting bar of the Agilent 4142B is connected between the CIRCUIT COMMON and CHASSIS GROUND terminals on the front of the GNDU plug-in.

If you cannot implement a floating-ground configuration, it may be necessary to open the Agilent 4142B shorting bar and connect the DUT ground to the circuit common grounds at the DUT ends of the SMU and GNDU cables. The circuit common grounds are not connected through the bias networks: they are available at the input jacks. Refer to the *Agilent 4142B Operation Manual* chapter 3, "Test Device Connections," and read the section titled "If the Test Device Is Externally Grounded (Floating Measurement)." This provides detailed instructions on making the necessary connections.

WARNING   A potential shock hazard exists when the shorting bar is disconnected for floating measurements. Do not touch any of the Agilent 4142B front panel connectors while a floating-ground measurement is in progress.

Configuring the SMUs and Bias Networks

Bias current and voltage are supplied to a device under test from plug-in source/monitor units (SMUs) in the Agilent 4142 DC source/monitor, connected to the device through the bias networks and probes or fixture. The standard Agilent 85123A system includes one medium-power SMU and one high-power SMU.

The quadraxial cables from the SMUs are routed through a cable feedthrough panel to the rear of the system rack, and out to the front of the rack above the work surface through another feedthrough panel.

Connect the cables emerging from the upper feedthrough panel to the bias networks, as explained in the next paragraphs.

Bias Connections

The following figure illustrates the connections from the SMUs to the bias networks. Connect the FORCE and SENSE connectors on the quadraxial cables to the FORCE and SENSE connectors on the bias networks. Connect the triaxial cable from the GNDU SMU to the ground connector on one of the bias networks (usually port 2). Leave the ground connector of the other bias network unconnected.
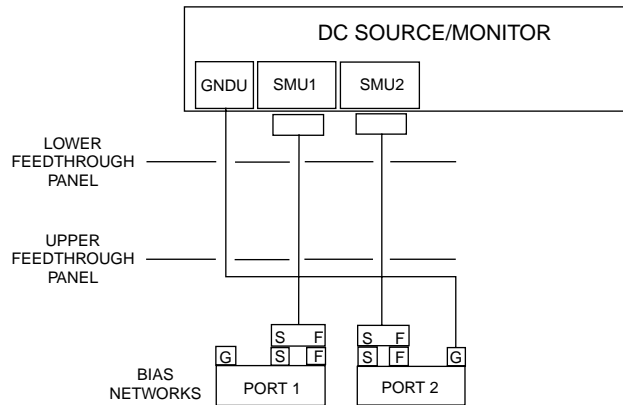
Figure 198   Connections from DC Source/Monitor to Bias Networks

Connections from Bias Networks to Device

<table>
<tr><td>NOTE</td><td>For in-fixture measurements, the measurement procedures in this manual apply to a FET mounted in the fixture in a common-source configuration, or a BJT mounted in a common-emitter configuration.</td></tr>
</table>

Connect the DC/RF signals from the outputs of the bias networks via the semi-rigid cables to the corresponding terminals of the fixture or probe station. Set up the angle of the cables and bias networks to minimize stress and torque, and ensure that the cables are properly supported. Iterate the positioning of the cables and bias networks until the connectors line up correctly. Use a torque wrench to tighten the connections.

The following figure illustrates the connections for a FET. The connections for a BJT are comparable, with the port 1 DC/RF signal connected to the base, the port 2 DC/RF signal connected to the collector, and the emitter grounded.
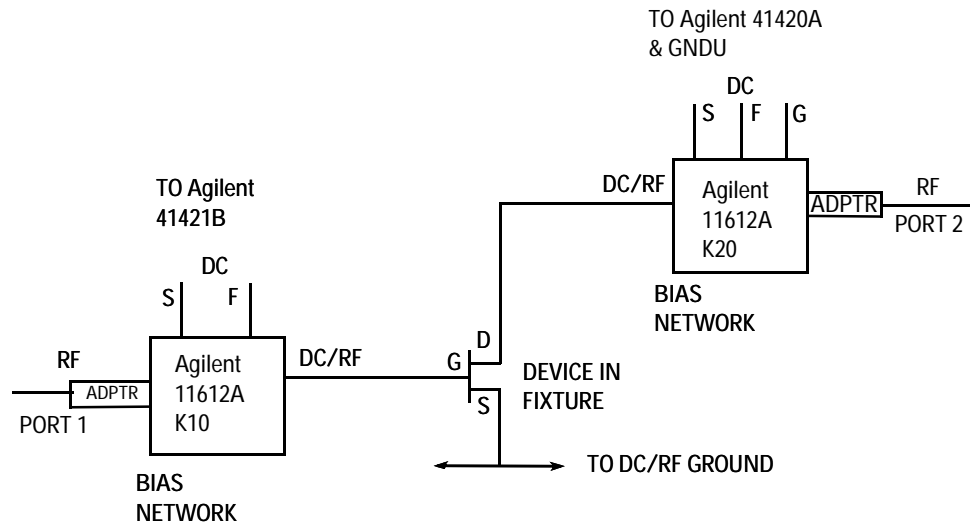
Figure 199   Connections from Bias Networks to In-Fixture DUT

- For additional hardware information, refer also to the installation and user's guide for your Agilent 85123A RF parameter extraction test system, and to the manuals for the individual instruments in the system.

- For information on fixtures or probe stations, refer to the manufacturer's documentation.

Switching on Power to the System

Be sure to switch on the network analyzer and the DC
source/monitor before the computer, so that the computer
recognizes the network analyzer subsystem and the DC
source/monitor.

Preparing for Calibration

Let the system warm up for at least 1 hour before calibrating the network analyzer. The DC measurements can be performed during this time.

The calibration standards will need to be at ambient room temperature when the measurement calibration is performed. If you are using a fixture, open the fixture calibration kit and place all the devices on top of the foam so they will reach room temperature by the time the system is warmed up. If you are using a probe station, place the probed impedance standard substrate (ISS) on the probe chuck and allow it to reach ambient temperature before use.

The calibration procedures can be found in the section "Swept Broadband Calibration" on page 632.

Terminating the Input Ports to Prevent Oscillation

Leave the network analyzer disconnected from the bias networks while you perform the DC measurements. Connect terminations with >20 dB return loss (such as 10 dB pads) to the RF IN connectors of the bias networks, to prevent possible bias oscillation. The network analyzer is calibrated later in the modeling procedures, immediately before the capacitance and S-parameter measurements.

Alternatively, isolate the device from RF power by manually setting 90 dB of attenuation at the network analyzer test ports. Also put the network analyzer in hold sweep mode by pressing MENU > TRIGGER MENU > HOLD.

Installing the Device

The best time to install the device is immediately before making measurements.

CAUTION   Ground yourself with an antistatic wrist strap to reduce the chance of electrostatic discharge. A ground terminal is provided on the lower left corner of the test set front panel for your convenience. (Note that the system work surface is coated with antistatic material and is connected to chassis ground, therefore a static mat is not required.)

Install the device carefully on the probe chuck or in the fixture. Be sure to handle the device as little as possible, to avoid damaging it. Once the device is in place, avoid bumping the test station.

NOTE   Eliminate direct light sources (such as microscope light) during measurement, especially for GaAs devices, as light may influence the measurement and cause inaccurate results.

Performing Hardware Setup in IC-CAP

The measurement system hardware is controlled by the IC-CAP software. This procedure configures IC-CAP to recognize the system instruments on the GPIB, and the individual source/monitor units (SMUs) in the DC source/monitor. The complete procedure need only be done after initial system installation, or any time the system hardware is changed, to establish a system handshake. The second part of the procedure (renaming the SMUs) also needs to be done any time you change device types, such as from FET to BJT or from BJT to FET.

For help with launching IC-CAP and opening the model file you need, refer to Chapter A, "Agilent 85190A IC-CAP" then continue with these steps:

1 In the IC-CAP Main window, select **Tools > Hardware Setup**. The hardware window is displayed. It will vary depending on the system hardware used, but will be similar to the following figure.
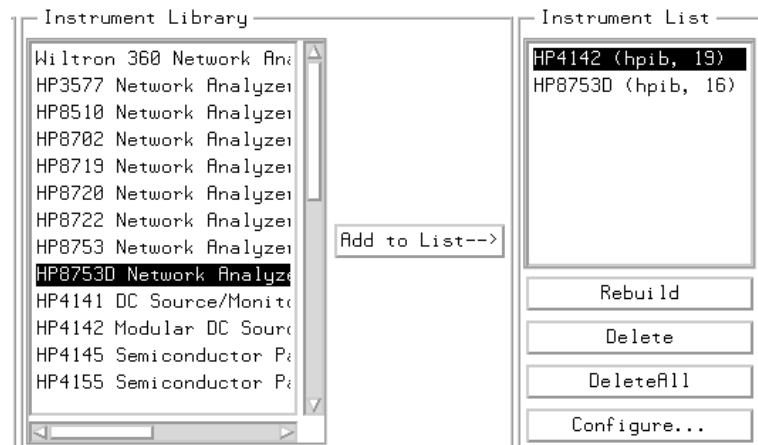
Figure 200   Hardware Window

2 Under the Instrument List select **Rebuild**. This command causes IC-CAP to poll all available GPIB addresses, and add to the Instrument List any newly connected instruments that

are powered up. (It does not affect instruments already on the list, whether they are powered up or not.)

Renaming the SMUs

Use this procedure any time you change device types, such as from FET to BJT or from BJT to FET, so that the measurement setups can properly communicate with the bias supplies.

When you run the IC-CAP software, it initially identifies the plug-in SMUs according to the numbers of the Agilent 4142 slots in which they are installed. A medium-power SMU occupies one slot. A high-power SMU occupies two slots, and is identified by the higher slot number of the two. An Agilent 85123 system can include different combinations of SMUs: the standard system includes two SMUs, one Agilent 41421B medium-power SMU and one Agilent 41420A high-power SMU. The medium-power SMU is factory-installed in slot 1 and is initially identified in the IC-CAP software as MPSMU1. The high-power SMU is installed in slots 2 and 3, initially identified as HPSMU3. The ground unit is identified as GNDU. It is generally more convenient to assign unit names to the SMUs that identify their purpose in a device measurement, depending on the bias connections at the device terminals and the type of device you are modeling.

In configuring IC-CAP to recognize the system hardware, you need to set the SMU names used in software to identify the SMU connections at the measurement terminals. In a FET measurement, the SMUs are identified as VG (gate supply) and VD (drain supply). In a BJT measurement, the SMUs are identified as SMU1 and SMU2, corresponding to the network analyzer test ports: this helps to facilitate measurement of a BJT with only two SMUs, since the port 2 SMU is used for measurements at both the collector and emitter terminals. Some models use more than two SMUs: their SMU names are defined in their respective chapters.

The following figure illustrates the SMU unit names for a FET measurement and the correspondence to the device terminal connections.
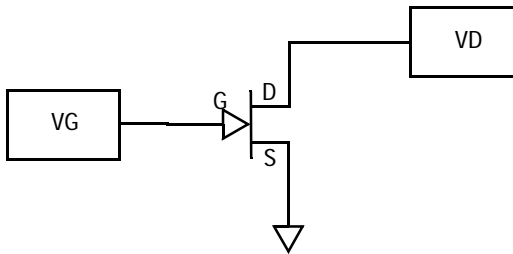
Figure 201   SMU Unit Names for a FET Measurement

The following figure illustrates the SMU unit names for a Gummel-Poon BJT measurement and the correspondence to the device terminal connections
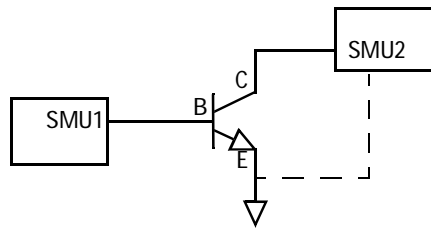
Figure 202   SMU Unit Names for a BJT Measurement

1 In the Instrument List click on **HP4142** to highlight it. Select **Configure** and the Configuration of HP4142 window is displayed.

2 The Unit Table lists the SMUs. Also listed for each is a unit ID (identification) name that can be edited.

  • In a FET measurement, the medium-power MPSMU1 is renamed VG, because it is connected to the gate terminal; and the high-power HPSMU3 is renamed VD, because it is connected to the drain terminal.

  • In a Gummel-Poon BJT measurement, the medium-power MPSMU1 is renamed SMU1 because it is the port 1 SMU; and the high-power HPSMU3 is renamed SMU2 because it is the port 2 SMU.

3 To make this change, click left with the mouse and move the mouse pointer over the name assigned to MPSMU1 to highlight it. Then type in the new name, and press Return. Similarly, change MPSMU3's assigned unit name. Then select **OK** to close the configuration window.

4 To save this hardware configuration in a file for future use, select **File > Save**. A file filter is displayed, showing the pathname of the current directory. Type in an appropriate name for the *.hdw* file you are creating, and select **OK**.

5 Close the hardware window.

After IC-CAP is configured to recognize the system hardware, you can proceed to one of the modeling chapters.

Defining the DC Source/Monitor Instrument State

NOTE   For details on IC-CAP instrument options (instrument states) for specific instruments, refer to Chapter 1, "Supported Instruments," in the *Reference* manual.

IC-CAP provides remote control of the DC source/monitor and the network analyzer to measure the device under test. This procedure configures IC-CAP with the instrument state settings (instrument options) for the DC source/monitor.

Use this procedure if you have been referred here from one of the device modeling procedures.

You will need to set the DC source/monitor instrument state separately for *each* DC or DC-biased measurement setup in a modeling procedure. However, use the same settings for all the measurement setups in one model (except that Integ Time can differ among setups). Unless you use the default settings documented here, it is helpful to write down the settings to duplicate in the other setups.

NOTE   DC signal levels and other input parameters that differ among measurement setups are not set in this procedure, but in the individual setup procedures in each model chapter.

The settings used in this procedure are defaults and the explanations are general.

1 Select the DC measurement setup of your choice.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states for the DC source/monitor. Figure 203 illustrates example DC source/monitor instrument states.

3 Set each of the DC source/monitor measurement parameters according to the actual Agilent 4142 configuration and the device to be measured, using the guidelines in the following steps.

Figure 203   Example Agilent 4142 DC Source/Monitor Instrument State

4 Set Use User Sweep to No, as it is unnecessary for these procedures, and the source/monitor internal sweep is faster.

5 Hold Time is the delay in seconds before starting a sweep, to allow for DC settling. Generally, no hold time is required.

6 Delay Time is the time in seconds the instrument waits before taking a measurement at each step of a sweep. Generally no delay time is needed.

7 For Integ Time, M (medium) is a good default choice. In measurements where a long integration time is needed for noise reduction, you would use L (long), and you will be instructed to do so in certain procedures. You can speed up a measurement by using S (short), but this is not recommended because it degrades the dynamic range of the measurement.

8 Set Range to 0 to implement SMU autoranging.

9 Power Compliance is used to set the maximum current/voltage combination for the DC source/monitor. However, in the models, SMU voltage and current compliances are set individually in the individual setups. Therefore the value here can be set to 0.000.

10 Set SMU Filters ON to `Yes`. This switches in low-pass filters on the SMU outputs, to protect the device from voltage spikes caused by DAC output changes.

11 High-frequency IC-CAP is not generally configured for pulsed measurements, therefore Pulse Unit can be left blank. All other Pulse settings are then irrelevant and can be ignored.

12 Module Control is not used in these procedures: leave the field blank.

13 Init Command sets the instrument to a mode not supported by other fields in this table. It is not generally used in these models. Leave the field blank unless instructed otherwise.

14 If you wish, refer to the DC source/monitor manual for more detail.

15 If the measurement setup you are configuring calls for a DC measurement only, close the instrument options window. Then return to the modeling procedure.

16 If the setup also calls for a network analyzer measurement, continue to "Defining the Instrument States for S-Parameter Measurements" on page 624.

Defining the Instrument States for S-Parameter Measurements

NOTE	For details on IC-CAP instrument options (instrument states) for specific instruments, refer to Chapter 1, "Supported Instruments," in the *Reference* manual.

IC-CAP provides remote control of the DC source/monitor and the network analyzer to measure the device under test. These procedures configure IC-CAP with both the DC source/monitor and the Agilent 8753 network analyzer instrument states for S-parameter measurements. The first procedure sets the instrument states for a broadband swept-frequency measurement, and the second for a CW measurement.

Instrument States for Swept Frequency Measurements

Use this procedure if you have been referred here from one of the device modeling chapters.

You will need to set the instrument states separately for *each* measurement setup in a modeling procedure. However, use the same settings for all the swept S-parameter measurement setups in one model (except that Integ Time can differ among setups). Unless you use the example settings documented here, write down the settings you use, to duplicate among setups within one model.

The settings used here are examples. Your settings may differ. Explanations here apply to all swept-frequency S-parameter measurements in the different device measurement procedures.

| NOTE | Source parameters such as frequency range are not set in this procedure, but in the individual S-parameter setup procedures in each model chapter. |

Do *not* use these settings for a CW S-parameter measurement. The settings for a CW measurement are slightly different, and are documented following this procedure.

1 Select the swept S-parameter setup of your choice.

2 Select the **Instrument Options** tab, and a window will be displayed showing the instrument states. The following figure shows an example.

```
— HP8753D.7.16 ——————————————
Use User Sweep              No
Hold Time                   0.000
Delay Time                  100.0m
Port 1 Source Power         -10.00
Port 1 Power Range[0-7]     0
Port 1 Auto Power Range      No
Coupled Port Power          Yes
Port 2 Source Power         -10.00
Port 2 Power Range[0-7]     0
Port 2 Auto Power Range      No
Power Slope                 0.000
Sweep Time                  100.0m
IF Bandwidth (AVG)          1000
Use Fast CW                  No
Avg Factor[1-999]           16
Cal Type[SHN]               H
Cal Set No [1..32]          1
Soft Cal Sequence           LOST
Delay for timeouts          0.000
Use Linear List              No
Init Command
```

Figure 204 Instrument States for Swept Frequency Measurements with Agilent 8753D

3 Set the Agilent 4142 instrument state the same as for the DC measurements.

4 Set the Agilent 8753 instrument state according to the actual hardware configuration, the calibration used, and the device to be measured, following these guidelines:

5 Set Use User Sweep to No, because this is a standard internal sweep.

6 Hold Time is the time in seconds before the instrument starts a sweep, to allow for DC settling. Generally, no hold time is required.

7 Delay Time is the delay in seconds the instrument waits before setting each frequency. No delay is needed here, but the default is 100 ms.

8 Port 1 Source Power and Port 2 Source Power are used with an analyzer that has an integrated test set. Set the power to the same level as in the network analyzer calibration. Take care that the power level will not be excessive at the device input or at the input port samplers of the analyzer. The default is -10 dBm.

9 Port 1 Power Range and Port 2 Power Range are used with an analyzer that has an integrated test set. The synthesized source in the analyzer contains a programmable step attenuator with eight power ranges. This lets you determine which range is used: the default is 0.

10 Port 1 and Port 2 Auto Power Range are used with an analyzer that has an integrated test set. These settings enable an autoranging power level and attenuation capability. The default IC-CAP setting is No.

11 Coupled Port Power is used with an analyzer that has an integrated test set. It couples port 2 to the port 1 power settings, ignoring the settings for port 2. The default is Yes, to couple the ports together.

12 If you calibrate with a power slope, set the corresponding value for Power Slope here, to better view the response of a device with power drop-off at higher frequencies. The units for power slope are dB/GHz; the default value is 0.000. (A network analyzer message will caution you that the correction may be invalid, but this can be ignored.)

13 Set the same Sweep Time as in the network analyzer calibration. You can check this on the network analyzer in

LOCAL mode by pressing MENU > SWEEP TIME. If you set a reduced IF bandwidth for calibration, the sweep time may have been slowed down automatically.

14 Set IF Bandwidth to the same value as in the calibration. The noise floor can be reduced by reducing the receiver input bandwidth.

15 Set Use Fast CW to No, because fast CW is not compatible with the instrument calibration.

16 Set Avg Factor to the same averaging factor you set in the calibration.

17 Set Cal Type to H (hardware) so that IC-CAP will recognize calibration device measurements at the network analyzer front panel.

18 Cal Set No must be set to the analyzer register number where you store your swept broadband calibration. Do not use register 6, which stores the active instrument state.

19 Soft Cal Sequence refers to the sequence of measurements of the cal standard devices: load-open-short-thru. You can set a different sequence if you prefer.

20 Delay for Timeouts increases the timeout and wait times. It can generally be set to the default value of 0.000.

21 If you define a specific number of points for the measurement calibration, set Use Linear List to No, the normal setting for a swept measurement. The setting would be Yes for a log or list sweep.

22 If necessary, refer to the "Application and Operation Concepts" chapter of the *Agilent 8753C Operating Manual* for more detail.

23 Close the instrument options window. Then return to the modeling procedure.

Different Settings for Agilent 8753 with External Test Set

An Agilent 8753 with an external test set uses different settings for power level and attenuation. Also, these analyzers do not have the power autoranging or coupled port power functions. These are the different settings:

1 Port 1 Atten and Port 2 Atten must be set to the same level of attenuation you set in the network analyzer calibration. The default is 20 dB.

2 Source Power must be set to the same level as in the network analyzer calibration. The default is −10 dBm.

Instrument States for CW Frequency Measurements

Use this procedure if you have been referred here from one of the device modeling chapters.

You will need to set the instrument states separately for *each* measurement setup in a modeling procedure. However, use the same settings for all the CW S-parameter measurement setups in one model (except that Integ Time can differ among setups). Unless you use the settings documented here, write down the settings you use, to duplicate among setups within one model.

The settings used here are examples. Your settings may differ. Explanations here apply to all CW S-parameter measurements in the different device measurement procedures.

1 Select the CW S-parameter setup of your choice.

2 Select the **Instrument Options** tab and a window will be displayed showing the instrument states. The following figure shows an example.

```
─ HP8753D.7.16 ──────────────
Use User Sweep          Yes
Hold Time               0.000
Delay Time              100.0m
Port 1 Source Power     -10.00
Port 1 Power Range[0-7] 0
Port 1 Auto Power Range No
Coupled Port Power      Yes
Port 2 Source Power     -10.00
Port 2 Power Range[0-7] 0
Port 2 Auto Power Range No
Power Slope             0.000
Sweep Time              100.0m
IF Bandwidth (AVG)      1000
Use Fast CW             No
Avg Factor[1-999]       16
Cal Type[SHN]           H
Cal Set No [1..32]      2
Soft Cal Sequence       LOST
Delay for timeouts      0.000
Use Linear List         No
Init Command
```

Figure 205   Instrument States for CW Frequency Measurements with
Agilent 8753D

3 Set the Agilent 4142 instrument states the same as for the DC
measurements.

> **NOTE** The network analyzer instrument state settings for a CW-frequency
> S-parameter measurement are the same as for a swept S-parameter
> measurement, with the following exceptions:

4 Set Use User Sweep to Yes because CW is a nonstandard
sweep.

5 Be sure to set the source power and attenuation levels the
same as in the network analyzer calibration.

6 Set Cal Set No to the analyzer register number where you stored your CW calibration. This must be different from the register used by the swept broadband calibration, and from any other CW calibration you make at a different frequency for any different measurement.

7 You can set Use Linear List to either Yes or No. This setting is ignored by IC-CAP for a CW measurement.

8 Close the instrument options window. Then return to the modeling procedure.

NOTE   For more information about IC-CAP instrument options (instrument states), refer to Chapter 1, "Supported Instruments," in the *Reference* manual.

Calibrating the Network Analyzer

The network analyzer must be calibrated before any S-parameter measurements are performed. The system must be allowed to warm up for at least 1 hour before calibration, and the calibration standards must be at ambient room temperature. It is a good practice to switch on the analyzer and expose the calibration devices to air before you make the DC measurements.

Good calibration of the network analyzer system is critical to a good high-frequency measurement and extraction. A good calibration is dependent on the quality of the calibration kit standard devices, the care with which they are maintained, and the correctness and repeatability of the device connections. A stable ambient temperature (±1°C) is required to maintain a good calibration.

For the high-frequency IC-CAP measurement procedures, the network analyzer is calibrated over a broadband frequency range, to perform the S-parameter measurements for AC and parasitic extractions. In addition, one or two calibrations may be needed at CW frequencies for specific measurement setups. These procedures explain how to set up a broadband calibration and a separate CW calibration. For the Agilent 8753 and 8720 systems, IC-CAP does not support the subset calibration technique used for an Agilent 8510 system.

The instructions given here are very abbreviated: more detailed information is available in the Agilent 8753 operating manual.

| NOTE | It is important that values entered in software correspond with actual instrument settings. Be sure to write down the parameters of your calibrations, so that they are readily available when you need to enter them in IC-CAP. |

To connect the network analyzer:

Disconnect the terminations from the bias networks. Connect the RF cables from the network analyzer test ports to the RF inputs of the bias networks.

Swept Broadband Calibration

1 In the IC-CAP model, select the S-parameter setup of your choice.

2 In the measurement setup, set the instrument options appropriately for your measurement. (See "Defining the Instrument States for S-Parameter Measurements" on page 624.)

3 You can set the network analyzer manually from its front panel, or use the IC-CAP Calibrate command in the setup menu to download the instrument options into the network analyzer. In either case, you perform the cal standards measurements at the analyzer front panel.

4 The following steps explain how to perform a manual network analyzer front panel calibration.

5 On the network analyzer, press LOCAL to gain front panel control. Press PRESET to return to a known standard state.

6 If you are using a system with the 6 GHz receiver option and you wish to measure in the 3 MHz to 6 GHz range, press SYSTEM > FREQ RANGE 3GHz6GHz.

7 To define the frequency range of the calibration and measurement, press START, and use the numerical keypad to set the start frequency, ending with one of the terminator keys (such as G/n) at the right of the keypad. Similarly, press STOP and set the stop frequency. Set a frequency range at least as wide as the operating range of the device.

8 Press MENU > NUMBER OF POINTS, and enter the number of points to be measured across the range. Allow enough points to provide good measurement resolution: at least 51 points are recommended.

9 Press MENU > TRIGGER MENU > CONTINUOUS to set a continuous stimulus sweep.

10 Press MENU > POWER, and set the desired power level. In setting the source power and test set attenuation, take care that the power level will not be excessive at the device input. Also consider the gain of the device, and set a power level that will not saturate the input port samplers of the analyzer. If a receiver input is overloaded (>+14 dBm), the analyzer automatically reduces the output power of the source to −85 dBm and displays the error message **OVERLOAD ON INPUT (R, A, B) POWER REDUCED**. In addition, the annotation P↓ appears in the left margin of the display to indicate that the power trip function has been activated. When this occurs, reset the power to a lower level, then toggle the SOURCE PWR on/OFF softkey to switch on the power again.

11 For a device with power dropoff at higher frequencies, you may wish to set a power slope using the SLOPE ON/off softkey. An appropriate power slope would be in the region of 2-3 dB/GHz.

12 Press AVG > AVERAGING FACTOR, and enter an averaging factor high enough to reduce trace noise and increase dynamic range as appropriate for your device measurements. A good default averaging factor is 256. To speed your measurements, you may find it convenient to set an averaging factor as low as 16. Press AVERAGING ON.

13 You can further reduce the noise floor by reducing the receiver input bandwidth. Press IF BW, and enter one of the following allowed values in Hz: 3000, 1000, 300, 100, 30, or 10. A tenfold reduction in IF bandwidth lowers the measurement noise floor by about 10 dB; however, the sweep time may be slower. For more information on averaging and the different trace noise reduction techniques, refer to the Agilent 8753 operating manual.

14 Press CAL > CAL KIT, and select the appropriate default or user-defined cal kit for your calibration devices.

15 If you wish to modify an internal calibration kit definition (see below), do so now.

16 Press CAL > CALIBRATE MENU, and perform the calibration of your choice at the analyzer front panel, measuring each of the standard devices in turn and pressing the softkeys as each measurement is complete. A full two-port cal provides

the greatest accuracy. A TRL* or LRM* cal is an appropriate alternative for in-fixture measurements. Omit isolation cal. Press DONE, and save the cal in the register number specified in the instrument options table. This must be a different register than you use for any CW cal.

17 Detailed procedures for measurements of calibration standards are provided in the Agilent 8753 operating manual.

CW Frequency Calibration

<table>
<tr><td>NOTE</td><td>A CW frequency calibration is similar to a broadband calibration with some exceptions. Repeat the procedure used for a broadband calibration, with these changes.</td></tr>
</table>

1 To set the CW frequency, press MENU > CW on the analyzer, and enter the frequency using the numerical keypad and one of the terminator keys.

2 Make a separate set of standards measurements.

3 When you save the calibration, use the cal register number specified in the instrument options table for this CW cal. This must be a different register than you used for the swept cal or any other CW cal.

<table>
<tr><td>NOTE</td><td>You can store instrument states and calibration data to an external disk, and restore them to the internal instrument registers as needed. Refer to "Memory Allocation" or "Saving Instrument States" in the Agilent 8753 operating manual for more information.</td></tr>
</table>

After the network analyzer is calibrated, return to the modeling chapter you are using for the S-parameter measurement procedures.

Using the Automatic IC-CAP Calibrate Function

Alternatively, you can use the IC-CAP Calibrate command for a broadband or CW calibration. This downloads the instrument states from IC-CAP to the network analyzer before you perform a calibration with the analyzer. This helps to avoid errors in matching values set in the analyzer and in IC-CAP.

1 In the IC-CAP model, select the S-parameter setup of your choice.

2 In the measurement setup, set the instrument options appropriately for your measurement.

3 From the setup menu select **Calibrate**. The instrument options are downloaded into the analyzer, together with the frequency stimulus values set in the Inputs.

4 The message **Calibrate HP8753.7.16 before measuring** is displayed. Select **OK** and perform the cal standards measurements from the analyzer front panel. Save the cal in the register specified in the instrument options.

Modifying a Cal Kit Definition

The network analyzer's internal calibration kit models are accurate for use with the standard coaxial calibration kits defined for use with the Agilent 8753. In certain circumstances, however, you may need a user-defined kit for compatibility with your calibration standards. Some possible cases might be these:

• If you are using a probe station and you know the probe capacitances, you can enter them into the cal kit definition so that their effects can be removed by the measurement calibration.

• The impedance of an on-wafer load may not be exactly 50Ω. You can characterize the DC impedance of a load exactly with a precision ohmmeter measurement, and enter the impedance value into the cal kit definition.

The procedure to modify an internal cal kit definition is complex, and not to be undertaken without a solid understanding of error correction and the system error model. It is explained in detail in *Product Note 8510-5, Specifying Calibration Standards for the Agilent 8510 Network Analyzer*

(Agilent literature number 5954-1559). For information specific to the Agilent 8753 network analyzer, including an example procedure, refer to the Agilent 8753 operating manual.

For information on the characteristics of calibration standards used with a fixture or probe station, refer to the manufacturer's device data sheets or other documentation.

TRL* Calibration Under IC-CAP Control

The Agilent 8753D network analyzer has the capability to perform a TRL* or LRM* calibration using internal firmware. TRL* (TRL-star) is an implementation of TRL that has been adapted for three-sampler network analyzers for use in fixtured measurement environments. (Note that this requires that you modify a user cal kit definition: see above.)

Earlier models of the Agilent 8753 do not have the internal TRL* calibration capability. The IC-CAP procedure described in the following pages allows you to perform a similar Agilent 8753C calibration using TRL cal standards compatible with your fixture. IC-CAP calculates the error coefficients and downloads them into the network analyzer. The procedure can be used for either a TRL* (thru-reflect-line) or TRM* (thru-reflect-match) calibration.

When the calibration is complete, the reference plane is defined at the middle of the thru standard, or at the interface to the DUT when it is installed in the compatible carrier.

For more information about TRL* calibration, refer to *Product Note 8720-2, In-fixture microstrip device measurements using TRL* calibration*, Agilent literature number 5091-1943E.

NOTE   The network analyzer GPIB address must be set to 16. It is recommended that you connect 10 dB pads at the RF IN connectors of both bias networks to improve the effective source and load match of the measurement setup.

The IC-CAP software for the TRL* calibration is in the model file *TRLCAL.mdl*. Find the TRL* calibration model under:

/examples/model_files/misc/TRLCAL.mdl

The message **WARNING: HP8753.7.16 Options saved in TRLCAL/cal/short** (or another filename) may be displayed. In this case, you will need to perform a Rebuild, as explained under "Performing Hardware Setup in IC-CAP" on page 617.

The TRL* Calibration DUTs and Setups

The following figure illustrates the DUTs and setups for *TRLCAL.mdl*.

The SETUPs are used to collect the measured raw S-parameters of various TRL (or TRM) standards, using variable values defined by you in the macros. IC-CAP computes the required error coefficients from this raw S-parameter data.
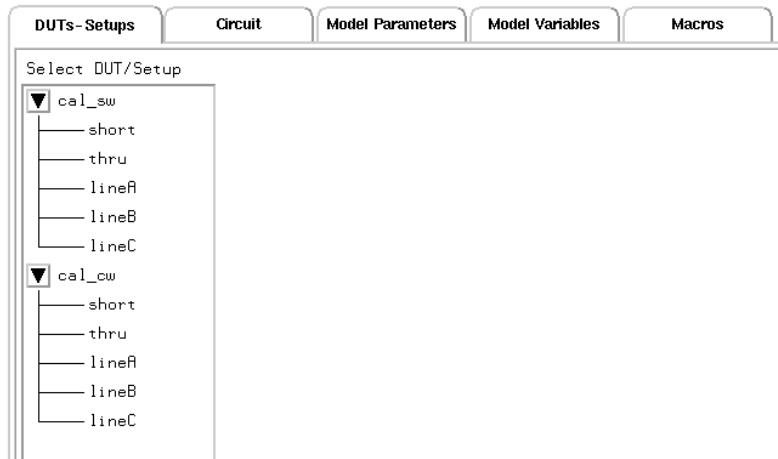
Figure 206   TRLCAL DUT/Setup Panel

The TRL* calibration uses separate setups for a swept frequency cal (cal_sw) and a CW frequency cal (cal_cw), which are both required in some modeling procedures. For the Agilent 8753 and 8720 systems, IC-CAP does not support the subset calibration technique used for an Agilent 8510 system, therefore

two sets of standards measurements are required. IC-CAP stores measured data in the setups for both the swept and CW cals.

The README macro provides an abbreviated set of instructions for the TRL* calibration procedure. The other macros provide interactive prompts for defining the instrument states and calibration frequencies, and for performing the calibration.

Definitions: Standards and Measurement Setups

The IC-CAP TRL* calibration requires measurements of three or more standard devices: a short, a thru, and at least one line standard (for TRL*) or match standard (for TRM*). The performance characteristics of the standards must be specified over the frequency range of the calibration. Therefore it may be necessary to use as many as three line standards to cover the total frequency range for a swept frequency cal. If so, you will need to define the transition frequencies between the line standards.

CAUTION   Some confusion can arise concerning the transition frequencies between the line standards, because the setup names do not necessarily correspond with the actual line standard names. Read the explanations in this and the following two pages to prevent possible problems in the way IC-CAP processes the calibration data.

Measurement Setups

The setups in the TRLCAL model measure the calibration standards, and store the measured raw S-parameter data in files with the same names as the setups. Each standard is measured over the full frequency range of the calibration. However, if multiple line standards are used, IC-CAP processes only the data delimited by the transition frequencies to compute the error coefficients of each. For a CW cal, IC-CAP processes only the data measured at the CW frequency you specify. The setups are used as follows:

Short: Measures the short standard and stores the measured data.

Thru: Measures the thru standard and stores the measured data.

LineA: Measures the first (lowest frequency) line standard and stores the measured data. In calculating the error coefficients for a swept frequency cal, IC-CAP uses only the data from the start frequency to the first transition frequency, or to the stop frequency if an above-range or zero-value first transition is defined. (You will define the calibration range and the transition frequencies later in the procedure.) The lineA standard can be represented either by a length of transmission line for a TRL* cal, or by a matched load for a TRM* cal.

LineB: Measures the second line standard (if any) and stores the measured data. In calculating the error coefficients for a swept frequency cal, IC-CAP uses only the data from the first transition frequency to the second transition frequency, or to the stop frequency if an above-range or zero-value second transition is defined. Typically the standard measured is a length of transmission line different from either the thru or lineA standards.

LineC: Measures the third line standard (if any) and stores the measured data. In calculating the error coefficients for a swept frequency cal, IC-CAP uses only the data from the second transition frequency to the stop frequency. Typically the standard measured is a length of transmission line different from either the thru, lineA, or lineB standards.

Transition Frequencies of Line Standards

Your calibration kit may have only one line (or match) standard, or it may have more than one line standard, each specified over a portion of the total specified cal kit frequency range. Refer to the data sheet for your calibration kit to determine the specified frequency ranges of each of the devices. For the RF frequency range you wish to measure, you may need to measure all the line standards, or you may need to measure only one or two. If

you measure more than one line standard, the specified upper frequency limit for the first line standard is the first transition frequency. Similarly, the specified upper frequency limit for the second line standard is the second transition frequency.

For example, if your total measurement frequency range is 300 kHz to 6 GHz and your first line standard is specified from 300 kHz to 1 GHz, then your first transition frequency is 1 GHz. If your second line standard is specified from 1 GHz to 5 GHz, then your second transition frequency is 5 GHz.

The *TRLCAL.mdl* file provides for measurements of up to three line standards for a swept frequency cal. If you use only one or two, you can define transition frequencies above the range of the calibration, and IC-CAP defaults to the start and/or stop frequencies. For example, if your total measurement frequency range is 300 kHz to 3 GHz and your second line standard is specified from 1 GHz to 5 GHz, you can set the second transition frequency to 5 GHz (or 0), and IC-CAP will measure the second line from 1 GHz to 3 GHz.

Measurement Setup Names vs. Actual Standards Names

The measurement setup names may not correspond with the names of the standards in your calibration kit. It is important to understand that the setup names are only filenames for the measured data. The actual fixtured standards you use for the line measurements will depend on the frequency range of your calibration, not on the names of the setups.

You *must* use the lineA setup for your lowest frequency line (or match) measurement, even if the cal standard you use is *not* line 1 in your calibration kit. This is because IC-CAP processes measured data from the setup files in the sequence they are shown in the model editor window. If a file is left empty (for example lineA), IC-CAP will stop processing data, even if there is measured data in lineB or lineC.

For example, you may have a calibration kit with a frequency range of 300 kHz to 6 GHz, and you may be calibrating for device measurements over a range of only 5 GHz to 6 GHz, using a short, a thru, and a single line (or match) standard. In this case,

you will use the lineA setup to measure your line or match standard that is specified for performance over the 5 GHz to 6 GHz frequency range, no matter how it is labeled.

If you are performing a CW cal alone, you *must* use the lineA setup for your line or match measurement.

Defining the IC-CAP TRL* Cal Instrument States

IC-CAP controls the network analyzer in measuring the calibration standards. These procedures configure IC-CAP with the required network analyzer instrument states to measure the TRL* calibration standards for the swept frequency and CW frequency cals for both the Agilent 8753D and earlier-model Agilent 8753s. (Instrument states are similar for Agilent 8720-family instruments, and are explained in Chapter 1, "Supported Instruments," in the IC-CAP *Reference* manual.) The Instr_opts macro lets you define the instrument state once for each cal and load it into all the corresponding measurement setups.

1 Select the **Macros** tab. Select the **Instr_opts** macro, and **Execute**. The instrument options window illustrated in the following

figure is for the short setup for a swept frequency cal using
an Agilent 8753D.

HP8753D.7.16	
Use User Sweep	No
Hold Time	0.000
Delay Time	100.0m
Port 1 Source Power	0.000
Port 1 Power Range[0-7]	0
Port 1 Auto Power Range	Yes
Coupled Port Power	Yes
Port 2 Source Power	0.000
Port 2 Power Range[0-7]	0
Port 2 Auto Power Range	Yes
Power Slope	0.000
Sweep Time	100.0m
IF Bandwidth (AVG)	1000
Use Fast CW	Yes
Avg Factor[1-999]	16
Cal Type[SHN]	N
Cal Set No [1..32]	cal_reg
Soft Cal Sequence	LOST
Delay for timeouts	0.000
Use Linear List	No
Init Command	FREQRANG3GHZ

Figure 207   TRL* Cal Instrument States for Swept Measurements with
Agilent 8753D

2 In the dialog box, select **OK** as prompted, then make any
instrument state changes required for your calibration and
measurements, using the illustrations and the following
instructions for guidance. The illustrated values are typical
defaults for a TRL* calibration used for measuring BJTs. The
settings you use may be different, depending on the
characteristics of the devices you intend to measure and
model.

Swept Measurements with Agilent 8753D

1 For swept measurements, set Use User Sweep to No because this is a standard internal sweep.

2 Set the source power level in dBm to 0.000 for measurements below 3 GHz. For measurements above 3 GHz, 20.00 is appropriate. The power levels are tied to the frequency range set in Init Command, below.

3 1000 Hz is a good default value for IF bandwidth. Reducing the IF bandwidth lowers the noise floor, at the cost of slowing the sweep speed. Allowed IF bandwidth values in hertz are 3000, 1000, 300, 100, 30, and 10.

4 Set Use Fast CW to Yes for a swept frequency measurement, to minimize repeated switching between the test set ports.

5 Cal Type (SHN) refers to *software/hardware/none*. Set it to N for "none." This is because the measurements you will make in this cal procedure are uncalibrated measurements. The finished calibration set will be used in your later measurement and modeling procedures.

6 Set Cal Set No to cal_reg, a variable used by the program.

7 The setting you use for Soft Cal Sequence is irrelevant, since Cal Type is set to None.

8 Set Use Linear List to No, otherwise the Agilent 8753 will be set to frequency list mode.

9 Set Init Command to FREQRANG3GHZ if you are using a standard Agilent 8753 network analyzer. (If you are using an Agilent 8753 with the 6 GHz extended frequency option and measuring above 3 GHz, enter FREQRANG6GHZ.)

Swept Measurements with Earlier-Model Agilent 8753

The window illustrated in the following figure is for the short setup for a swept frequency cal using an earlier-model Agilent 8753.

```
┌─ HP8753.7.16 ──────────────
│ Use User Sweep       │ No            │
│ Hold Time            │ 0.000         │
│ Delay Time           │ 100.0m        │
│ Port 1 Atten         │ 20.00         │
│ Port 2 Atten         │ 20.00         │
│ Source Power         │ 0.000         │
│ Power Slope          │ 0.000         │
│ Sweep Time           │ 100.0m        │
│ IF Bandwidth (AVG)   │ 1000          │
│ Use Fast CW          │ Yes           │
│ Avg Factor[1-999]    │ 16            │
│ Cal Type[SHN]        │ N             │
│ Cal Set No [1..6]    │ cal_reg       │
│ Soft Cal Sequence    │ LOST          │
│ Delay for timeouts   │ 0.000         │
│ Use Linear List      │ No            │
│ Freq Range[36N]      │ N             │
│ Init Command         │ FREQRANG6GHZ  │
```

Figure 208   TRL* Cal Instrument States for Swept Measurements with Earlier-Model Agilent 8753 (>3 GHz)

We recommend you use the same attenuator settings at port 1 and port 2, since the earlier Agilent 8753 analyzer uses the same attenuator for both ports. In setting power levels, you will need to set a source power sufficient to maintain phase lock, while taking care not to saturate the device output. Settings of 10 to 20 dB attenuation at both ports with a source power level of 0 dBm are appropriate for many devices with gain.

CW Measurements

1 Some of the instrument state settings for the TRL* CW cal are the same as for the swept frequency cal. The following settings are different, as illustrated:

Figure 209 TRL* Cal Instrument States for CW Measurements with Agilent 8753D

2 For CW measurements, set Use User Sweep to Yes.

3 You may not need to set attenuation for CW measurements, since gain may not be a factor. The CW measurements are usually used to determine the junction capacitance of a device. For this purpose the device is biased in such a way that it has loss rather than gain.

4 Set Use Fast CW to No.

Figure 210   TRL* Cal Instrument States for CW Measurements, Earlier Agilent 8753

5 When you have completed the settings, select the **Instr_dupl** macro, and **Execute**. This macro loads the short instrument options for the swept frequency cal into the other cal_sw setups; and the short options for the CW cal into the other cal_cw setups (thru, lineA, lineB, lineC).

> **NOTE**   When you execute the Instr_dupl macro, the macro must write files to disk. If the instrument options do not load down, your working directory probably does not have write permission set. To correct this, from your working directory type, `chmod 644 <directory_name>` and press Enter.

6 A message in the UNIX window will tell you when the instrument options are duplicated in all the setups. Ignore any **does not cascade** warnings.

Defining the Calibration Frequencies

You define the values for frequency range, number of measurement points, and transition frequencies between line standards in the Cal_freqs macro, as follows:

1 Select the **Cal_freqs** macro, and **Execute**. A dialog box will be displayed.

2 Enter the start frequency for your swept frequency calibration as prompted, and select **OK**.

3 Enter the stop frequency and select **OK**. The stop frequency must be greater than the start frequency.

4 At the prompt to enter the number of frequency points, use one of the following Agilent 8753 allowable values:

3	11	26	51
101	201	401	801
1601			

5 If you are doing a CW cal along with the swept cal, enter the CW frequency. If not, enter 0. (If you are doing a CW cal alone, refer to the *Note* below.) Select **OK**.

6 If you are using more than one line/match standard, enter the transition frequency for the first line (between lineA and lineB). Select **OK**.

7 If you are using three line standards, enter the transition frequency for the second line (between lineB and lineC). Select **OK**.

8 If you are using only one or two line standards, enter above-range or zero-value transition frequencies for the setups that do not apply. IC-CAP will measure only from the start to the stop frequency.

Refer to the paragraphs under "Definitions: Standards and Measurement Setups" on page 638 for explanations of transition frequencies and the difference between the

software setup names and the hardware standards names. The information there can prevent possible problems in the way IC-CAP processes the data.

9 When you have defined all the calibration frequencies, the UNIX window will list them, and will conclude with the message **Completed**.

<table><tr><td>**NOTE**</td><td>If you are doing a CW cal alone (instead of with a swept cal), enter a value of 0 to the prompts for start frequency, stop frequency, and number of points. Remember to use the lineA setup for your line/match measurement. The Calibration macro will subsequently provide only prompts that apply to a CW cal.</td></tr></table>

Performing the Calibration

The Calibration macro provides a series of prompts to guide you through the standards measurements. In this part of the procedure, you follow the prompts to connect the defined standards, and instruct IC-CAP to measure them. You then specify network analyzer registers to store the swept and CW calibrations, and IC-CAP calculates the error coefficients from the measured raw S-parameter data, then downloads them into the analyzer registers.

1 Select the **Calibration** macro, and **Execute**.

2 IC-CAP displays a dialog box, prompting you to perform the standards measurements you have defined. In each case, *first* connect the standard, *then* respond to the prompt by entering s for short, t for thru, A for the lineA setup, B for the lineB setup, and C for the lineC setup. Each time you enter a single letter, press Return. A message in the UNIX window tells you the standard is measured.

<table><tr><td>**CAUTION**</td><td>Be sure to connect the calibration devices that correspond with the frequencies you defined in the Cal_freqs macro. Otherwise your calibration may be compromised. For more information, refer to "Definitions: Standards and Measurement Setups" on page 638.</td></tr></table>

3 Following this prompt sequence, a dialog box asks you **Done? [y/n]**. If all the necessary cal standards have been measured, type y and select **OK**. (If you want to remeasure a standard, type n, reconnect the device, and type the appropriate character for the device setup. IC-CAP will remeasure, and replace the old data with the new.)

4 The next dialog box asks you to enter the calibration storage register for the swept cal. Type in a number between 1 and 4, and select **OK**. (Do not use register 5, because IC-CAP reserves it for temporary storage.) IC-CAP calculates the error coefficients, downloads them into the network analyzer, and turns on the analyzer calibration.

5 Next enter a different register number for the CW calibration, and select **OK**. IC-CAP performs the CW cal and downloads it.

6 The instrument state and calibration are saved in the analyzer's short-term non-volatile memory (that is, even if the instrument is switched off for as long as 72 hours).

7 Whenever you use these swept and CW TRL* calibrations for a modeling procedure, enter the corresponding register numbers in the Cal Set No field of the IC-CAP instrument options window. Also set the Cal Type (SHN) field to H for hardware.

If you wish to verify the calibration, you can return the network analyzer to local operation and make a calibrated thru measurement.

Saving the Calibration Data in IC-CAP

When the calibration is complete, save the model file with the measured standards data in it, so that you can always quickly retrieve the error coefficients and re-enter them into the network analyzer:

1 Select **File > Save As**.

2 In the dialog box that appears, type in an appropriate pathname and filename. Examples of possible filenames might include the calibration date or frequency, such as *TRLCAL_101597.mdl* or *TRLCAL_6GHz.mdl*. Select **OK**. First the CW cal is saved in one analyzer register using the variable

cal_reg_cw; then the swept cal is saved in another register using the variable cal_reg_sw.

3 When you want to retrieve the error coefficients at some future time, read in the *.mdl* file you saved. Select the **thru** setup for either the swept or CW cal. Make sure the Agilent 8753 instrument state corresponds exactly to the instrument options in the thru setup. Select the **CalHP8753** transform, and **Execute**. IC-CAP will calculate the error coefficients, download them into the network analyzer, and turn on the analyzer calibration.

Press LOCAL on the analyzer to return it to front panel control, then press SAVE and a register softkey to save the cal in one of the analyzer's memory registers.

This ends the procedures for TRL* calibration under IC-CAP control.

D

Using IC-CAP Data with MDS

This appendix provides information on using IC-CAP model data in the Agilent RF and Microwave Design System (MDS).

Saving Model Data

It is important that model data is saved correctly at the end of a parameter extraction or model generation procedure. Refer to the information on storing the model provided at the end of each of the model chapters. Note that the procedures are not necessarily the same in all cases.

Reading IC-CAP Data into MDS

Measured, simulated, and transformed IC-CAP data can be read into datasets in the Agilent RF and Microwave Design System (MDS). The standard compatible files are model, DUT, and setup files suffixed *.mdl*, *.dut*, and *.set*. However, these suffixes are not required: if the data is in the correct format, any file name or extension can be used.

When you are using MDS and wish to access an IC-CAP model, follow these steps:

1 Make the screen environment, a file icon, or any other collector environment active.

2 Use the menu command **INSERT > DATASET** and the mouse to insert a dataset icon. Rename the icon if desired.

3 Open the dataset icon.

4 When the index page appears, choose the menu command **PERFORM > READ > ICCAP FILE** (see the following figure).
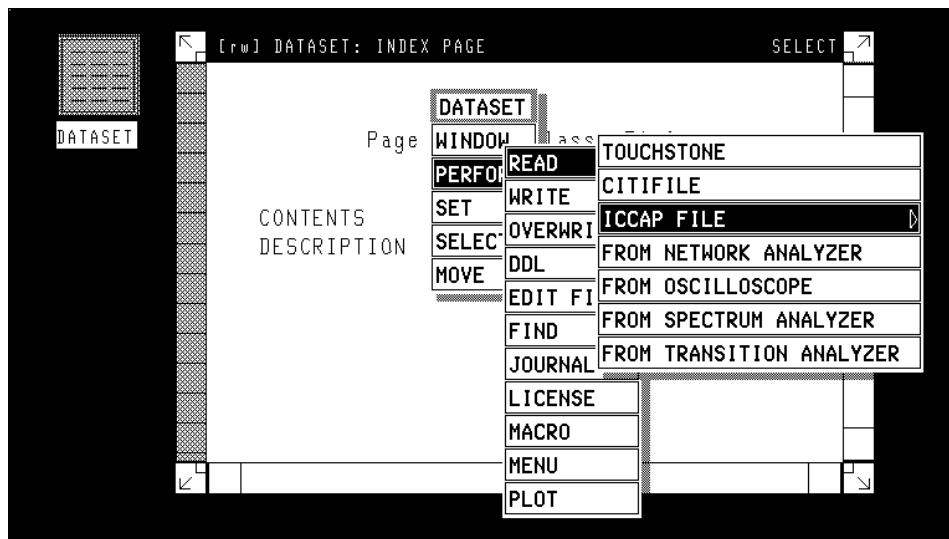
Figure 211   Reading IC-CAP Data

5 When the message appears at the top of the screen, type the name of the file that contains the IC-CAP data. If the data file is not in the current directory, type the full path name as well as the file name. Press Enter.

When the IC-CAP data file has been read, data packages are listed on the index page of the dataset. These correspond to setup names in the IC-CAP data file. The following figure shows an example.

Figure 212   IC-CAP Data Packages in MDS Dataset

Translating IC-CAP Data Using iccap2citi

IC-CAP data files can be translated into HP CITIfile format using the program iccap2citi in the UNIX operating system environment. This is often convenient when many files are to be translated.

The program is located in:

$MDSROOT/hp85150/bin/iccap2citi

IC-CAP includes the most current version of the iccap2citi program in:

$ICCAP_ROOT/bin/iccap2citi

This version should be copied to the MDS directory, or be referenced via the MDS aliasing mechanism as described in the MDS documentation.

```
.iccap2citi
usage: iccap2citi [-msx] [file1] [file2]
or: iccap2citi [-ivmsx] file1 file2
-p output model parameters only
-l output model parameters only, in lib format
-m skip MEAS data
-s skip SIMU data
-x skip XFORM data
-i interactively select data
-v print names of all packages translated
```

On the command line, *file1* is the file to be translated (the input file), and *file2* is the file that results (the output file). These names are optional, however.

If no name is specified for the output file *file2*, the output goes to standard out. If neither an input file name *file1* nor an output file name *file2* is specified, the input is taken from standard in and the output is sent to standard out.

The file that results is in HP CITIfile format, and can be read into a dataset in MDS using the menu command **PERFORM > READ > CITIFILE** on the index page in the dataset.

Reading Extracted Parameters from IC-CAP to MDS

You can access the individual model parameters saved in a
SPICE deck, and read them directly into device models in the
Agilent RF and Microwave Design System. This is done using
the DDL script *geticcap.ddl*. Note that the model parameters
must have been saved using the IC-CAP Save Extracted Deck
function. Follow these steps:

1   On the MDS circuit page, insert the model or model form
    element: for example, BJTMODEL or BJTMODELFORM.

2   Choose the menu command PERFORM/DDL.

3   When the message appears asking for the name of the DDL
    command file (script), type `geticcap.ddl`. Then press
    Enter.

4   When the next message appears, type the name of the IC-CAP
    file that contains the extracted parameter values (the SPICE
    deck for the desired model). Press Enter.

5   When the next message appears, type the name of the model
    or model form component into which the extracted IC-CAP
    values will be read. For example, type `CMP2`. Press Enter.

The IC-CAP parameters will be read in and the model
identification field MODEL will have the name that is used as
the subcircuit name in the parameter file.

If the collector mounting capacitance appears in the parameter
file, it is reported but is not used.

Special Considerations in Reading MOSFET SPICE Parameters

Special considerations must be taken into account in reading the parameters from a SPICE deck using *geticcap.ddl*, for two reasons. First, the extrinsic parasitic elements are not part of the MOSFET device or model card and must be added by the user. Second, only one device can be inserted into MDS at a time, but the scalable level 3 MOS model measures three devices. Explanations follow.

Adding Extrinsic Parasitic Elements

The extrinsic parasitic elements RG, LGG, LDD, and LSS that appear below the .SUBCKT line in the SPICE deck are not part of the MOSFET device or of the model card. (The intrinsic resistance elements RS and RD are part of both the device and the model card.) In order to include these elements as part of your MDS circuit, you need to add them as separate devices connected appropriately to the model element, using the extracted values given in the SPICE deck.

Setting the Geometry Parameters

In addition to the extrinsic parasitics listed above, certain device geometry parameters listed above the *.MODEL* line in the deck are not part of the model card. The geometry parameters L, W, AD, PD, AS, and PS must be entered for the specific device you intend to simulate. In a scalable MOSFET extraction, three devices are measured (large, short, and narrow). However, when you insert a device into MDS you will use only one of the devices measured in the extraction, or a device of a different size. Decide which device you will simulate and enter the corresponding geometry values for that device into the deck, using PERFORM > EDIT COMPONENT. (If you use the nonscalable MOS model with measurements of only one device, you will of course use the geometry parameters for that device.

These are the descriptions and default values for the geometry parameters:

Table 53   MOSFET Level 3 Geometry Parameters

Parameter	Description	Default Value
L	Channel length in meters.	100 μm
W	Channel width in meters.	100 μm
AD	Area of drain diffusion in square meters.	0.0
AS	Area of source diffusion in square meters.	0.0
PD	Perimeter of drain junction in meters.	0.0
PS	Perimeter of source junction in meters.	0.0

AS, PD, and PS are used in the equations describing the junctions. The reverse current can be input either as IS (in A) or as JS in (A/m2). Whereas the first is an absolute value, the second is multiplied by AD and AS to give the reverse current of the drain and source junctions respectively. The same idea applies also to the zero-bias junction capacitances CBD and CBS (in F) on the one hand, and CJ (in F/m2) on the other. PD and PS multiply CJSW (in F/m) to obtain the perimeter capacitance.

Example SPICE Deck for MOSFET Level 3

The following is an example of an extracted SPICE deck for a high-frequency MOSFET level 3 model:

```
.SUBCKT mos3_nhfs 1 2 3 4
RG 22 12 7.061
LGG 2 22 1.694E—10
LDD 1 11 3.538E—10
LSS 3 13 3.591E—11
M1 11 12 13 4
+ NMOS
+ L = 1.5E—06
+ W = 0.000352
+ AD = 1.936E—09
+ PD = 0.000176
+ AS = 1.936E—09
+ PS = 0.000176
.MODEL NMOS NMOS
+ LEVEL = 3
+ NSUB = 1.898E+16
+ UO = 1390
+ VTO = 0.5171
+ TPG = 1
+ NFS = 7.547E+12
```

```
+ TOX = 2.063E—08
+ NSS = 1E+10
+ VMAX = 1.546E+05
+ RS = 4.22
+ RD = 11.8
+ CBD = 5.518E—13
+ CJ = 0.000285
+ MJ = 0.2024
+ CJSW = 1.291E—15
+ MJSW = 0.1
+ IS = 1E—14
+ PB = 0.2972
+ FC = 0.5
+ XJ = 2.675E—07
+ LD = 1E—09
+ DELTA = 0.002
+ THETA = 1E—06
+ ETA = 0.2
+ KAPPA = 1E—06
+ CGSO = 2.113E—10
+ CGDO = 1.674E—10
+ CGBO = 0
+ XQC = 1
.ENDS
```

Creating Devices in MDS Using IC-CAP Parameters

You can use extracted IC-CAP parameter values to create devices in the Agilent RF and Microwave Design System, using the DDL script *addiccap.ddl*. Follow these steps:

1 In MDS, insert and open a file icon for the device to be created.

2 Open the file icon for the system microwave library mwlib and in it open the partition COMPONENTS and, in turn, the partition mns_only.

3 Use the menu command **COPY** and the mouse to copy the design icon for the device from the partition mns_only to the new file icon window. For example, copy the design icon BJTMODEL or BJTMODELFORM.

4 Rename this design icon, to avoid creating two devices with the same name. For example, add a number to the name, as in BJTMODEL8.

5 In the file icon window, choose the menu command **PERFORM > DDL**.

6 When the message appears asking for the name of the DDL command file (script), type addiccap.ddl. Then press Enter.

7 When the next message appears, type the name of the IC-CAP file that contains the extracted parameter values (the SPICE deck). Press Enter.

8 When the next message appears, type the name of the model or model form component into which the extracted IC-CAP values will be read. For example, type BJTMODEL8. Press Enter.

The IC-CAP parameters will be used to create a scion of the IC-CAP device in the MNS design icon, using the filename of the IC-CAP parameters. Devices with this name (and the individual device parameters) can now be used like other MDS model elements to create circuit schematics.

Additional MDS Information

For detailed information on the functions of the Agilent RF and Microwave Design System, refer to its documentation.

D Using IC-CAP Data with MDS

Index

A

A parameters, extracting, 106
AC parameters, extracting, 109
AC_measure macro, 415, 417, 429

B

base resistance
    BJT, 133, 135, 161, 187
    EEBJT2, 404, 414, 416, 430, 447
beta parameters, extracting, 108
bias networks
    connections, 575, 579, 580, 590, 609, 611, 612, 613
    floating ground, 611
    oscillation prevention, 583, 615
    overview, 573, 578, 607
    safety considerations, 578, 610
    terminating, 583, 615
BJT, high-frequency, model generator, 115

C

calibration
    85122A
      cal kit definitions, modifying, 592
      CW frequency subsets, 591
      overview, 589
      preparation, 582
      procedure, swept broadband, 590
    85123A
      cal kit definition, modifying, 635
      preparation, 615
      procedure, 631
      procedure, CW frequency, 634
      procedure, swept broadband, 632
    automatic, 635
    frequency definition, 647
    frequency macro, 647, 648
    saving cal data, 649
    TRL

defining instrument state, 641
    definitions, 638
    DUTs and Setups, 637
    line standards transition frequencies, 639
    measurement setups, 638
    naming conventions, 640
    overview, 636
capacitance parameters
    extracting, 109
channel resistance, 39, 95, 307
collector resistance
    BJT, 133, 134, 159
    EEBJT2, 404
contact resistance, 340, 347, 351, 377, 390
control
    variable flags, 312
    variables vs. variable tables, 313

D

Data_sheet_vars macro, 408, 409, 418, 429
diode
    Agilent root, model generator, 247
    resistance, 276
drain resistance, 88
dynamic resistance, 318, 340

E

EEBJT2, model generator, 399
EEFET3/EEHEMT1, model generator, 299
emitter resistance
    BJT, 135, 136, 160
    EEBJT2, 404, 414, 416, 448
end resistance, 336
environment variables, setting, 15, 199, 250
Extract_DC macro, 414, 418, 425, 439
Extract_extrin macro, 412, 416

Extract_junc_cap macro, 416
Extract_transit macro, 416

F

FET
    Agilent root, model generator, 11
    high-frequency, model generator, 69
file types, model, 295
flags, control variable, 312
floating ground, 578

G

gamma parameters
    extracting, 108
gate
    diode resistance, 39
    resistance, 88, 390

I

IC-CAP
    framework, 564
    using MDS with, 196
ideality factor
    extracting, 41, 103, 278, 279
    measuring, 39, 84, 276
initialize parameters macro
    root FET, 20, 31, 40, 44, 55, 61
    root MOSFET, 203, 204, 242
initialize values macro, 254, 255
instrument options macro, 646
instrument state
    definition
      DC source/monitor, 621
      network analyzer, 624
    downloading from IC-CAP, 635
    TRL calibration, 641
interlock connection, 578
intrinsic output resistance, 77, 108