



**Agilent Technologies**

ADS 2008  
January 2008  
DesignGuide Utilities

# Advanced Design System 2008

© Agilent Technologies, Inc. 2000-2008

395 Page Mill Road, Palo Alto, CA 94304 U.S.A.

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

## Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Microsoft®, Windows®, MS Windows®, Windows NT®, and MS-DOS® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Java™ is a U.S. trademark of Sun Microsystems, Inc. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC.

**Errata** The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

**Warranty** The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

**Technology Licenses** The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license. Portions of this product include the SystemC software licensed under Open Source terms, which are available for download at <http://systemc.org/>. This software is redistributed by Agilent. The Contributors of the SystemC software provide this software "as is" and offer no warranty of any kind, express or implied, including without limitation warranties or conditions or title and non-infringement, and implied warranties or conditions merchantability and fitness for a particular purpose. Contributors shall not be liable for any damages of any kind including without limitation direct, indirect, special, incidental and consequential damages, such as lost profits. Any provisions that differ from this disclaimer are offered by Agilent only.

**Restricted Rights Legend** If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

## Contents

- Transistor Bias Utility
- - Step-by-Step Example
  - Setting Up the Design Environment
    - - Setting DesignGuide Preferences
    - - Opening a Project
    - - Opening a Schematic Window
    - - Opening the Transistor Bias Utility
    - - Using the Control Window
  - Designing and Analyzing a Network
    - - Displaying the SmartComponent Palette
    - - Placing Example Component in the Design
      - Wire the Gate, Drain, and Source in the Example.
    - - Changing SmartComponent Parameters
    - - Designing the SmartComponent
    - - Closing the FETBias Analysis Results Window
    - - Examining the Component Design
    - - Deleting the SmartComponent
  - Using SmartComponents
    - - Placing and Editing SmartComponents
    - - Placing SmartComponents
    - - Changing Position and Orientation
    - - Editing SmartComponents
      - The FET Bias Network Component
    - - Copying SmartComponents
      - Copying Within a Design
      - Copying Between Designs or Schematic Windows
      - Copying a SmartComponent as a Unique Design
    - - Deleting SmartComponents
      - Deleting from Current Design
      - Deleting from Current Project
      - Deleting Manually Using File System
  - Using SmartComponents as Standalone Components
    - - Using an Existing SmartComponent Within the Same Project
    - - Using an Existing SmartComponent in Any Project
  - Automated Design and Analysis
  - Resistive Networks
    - - Resistive Bias Network SmartComponents
      - Appropriate Design Setup for Resistive Bias Network
    - - BJT Networks
    - - FET Networks
    - - Bias Point Selection for Resistive Bias Networks
  - Active Networks
    - - Active Bias Network SmartComponents
      - Appropriate Design Setup for Active Bias Networks
    - - Network Design
    - - Regulating Networks

- Bias Point Selection for Active Bias Networks
- Load Pull Measurement Data Import Utility
  - Importing Loadpull Measurement Data
  - Preparing Data for Interpolation
  - Interpolating the Data and Displaying Results

## Transistor Bias Utility


The Transistor Bias Utility documentation provides an introduction to the Transistor Bias Utility. The complexity of the Advanced Design System (ADS) is made easily accessible through the automated capability. A first-time or casual ADS user can begin using the capability of ADS quickly, while experienced ADS users can perform tasks faster than ever before. The [Step-by-Step Example](#) describes how a resistive bias network for a GaAs FET can be designed and verified in a few minutes.

The Transistor Bias Utility provides SmartComponents and automated-assistants for the design and simulation of common resistive and active transistor bias networks. The automated capabilities can determine the transistor DC parameters, design an appropriate network to achieve a given bias point, and simulate and display the achieved performance. All SmartComponents can be modified. You simply select a SmartComponent and, with little effort, redesign or verify their performance. [Using SmartComponents](#) provides details about using SmartComponents.

## Step-by-Step Example

The step-by-step example takes you through the design and analysis for a resistive bias network for a GaAs FET. After completing this example, you should have a basic understanding of the Utility and be ready to begin using it. Follow these steps to begin:

- [Setting Up the Design Environment](#)
- [Designing and Analyzing a Network](#)

 **Note**  
You should already be familiar with the basic features of Advanced Design System. For help with ADS basic features, refer to the [Schematic Capture and Layout](#) documentation.

## Setting Up the Design Environment

Before you can use the Transistor Bias Utility, you must set up the design environment by using these steps:

- [Setting DesignGuide Preferences](#)
- [Opening a Project](#)
- [Opening a Schematic Window](#)
- [Opening the Transistor Bias Utility](#)
- [Displaying the SmartComponent Palette.](#)

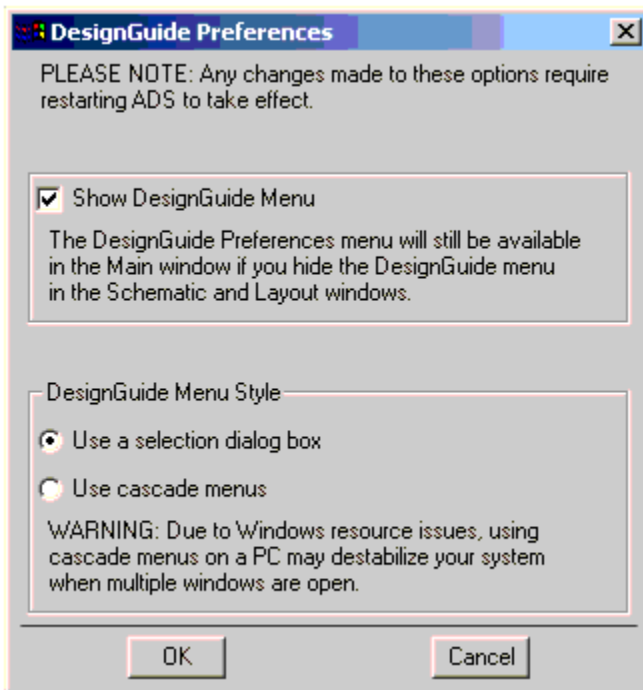
**Note**  
Before beginning, you must have installed the DesignGuide with appropriate licensing codewords.

## Setting DesignGuide Preferences

All DesignGuides can be accessed through either cascading menus or dialog boxes. You can configure your preferred method in the ADS Main window or from the Schematic window.

To configure access through menus or dialog boxes:

1. From the Main or Schematic window, choose DesignGuide > Preferences.
2. In the DesignGuide Menu Style group box, choose either Use a selection dialog box or Use cascade menus.



3. Close and restart the program for your preference changes to take effect.

**Note**  
On PC systems, Windows resource issues might limit the use of cascading menus. When multiple windows are open, your system could become destabilized. Therefore, the dialog box menu style might be best for these situations.

The ADS Main window DesignGuide menu contains these choices:

DesignGuide Developer Studio > Start DesignGuide Studio is only available on this menu if you have installed the DesignGuide Developer Studio to open the initial Developer Studio dialog box.

DesignGuide Developer Studio > Developer Studio Documentation is only available on this menu if you have installed the DesignGuide Developer Studio to open the DesignGuide Developer Studio documentation.

**Note**  
Another way to access the DesignGuide Developer Studio documentation is by selecting Help > Topics and Index > DesignGuides > DesignGuide Developer Studio from any ADS program window.

Add DesignGuide opens a directory browser in which you can add a DesignGuide to your installation. This is primarily intended for use with DesignGuides that are custom-built through the Developer Studio.

List/Remove DesignGuide opens a list of your installed DesignGuides. Select any that you would like to uninstall and choose the Remove button.

Preferences opens a dialog box that enables you to:

- Disable the DesignGuide menu commands (all except Preferences) in the Main window by unchecking this box. In the Schematic and Layout windows, the complete DesignGuide menu and all of its commands are removed if this box is unchecked.
- Select your preferred interface method, either cascading menus or dialog boxes.

### Opening a Project

The ADS design environment is set up within a project.

To create a new project:

1. From the ADS Main window, choose File > New Project or click Create a New Project on the toolbar.



2. In the dialog, define the location of the project and assign a project name.

### Opening a Schematic Window

A new schematic design is needed to contain the lowpass component for this example.

To open a Schematic window:

1. From the ADS Main window, choose Window > New Schematic or click New Schematic Window on the toolbar. A new Schematic window appears.



- ✔ Hint  
Depending on how your ADS preferences are set, a Schematic window can appear automatically when you create or open a project.

2. In the Schematic window, choose File > New Design to create a design named Example.

### Opening the Transistor Bias Utility

The Transistor Bias Utility is accessed from the DesignGuide menu.  
To open the Transistor Bias Utility:

1. In the Schematic window, choose one of these paths from the DesignGuide menu:
  - DesignGuide > Amplifier > Tools > Transistor Bias Utility
  - DesignGuide > Mixers > Tools > Transistor Bias Utility
  - DesignGuide > Oscillator > Tools > Transistor Bias

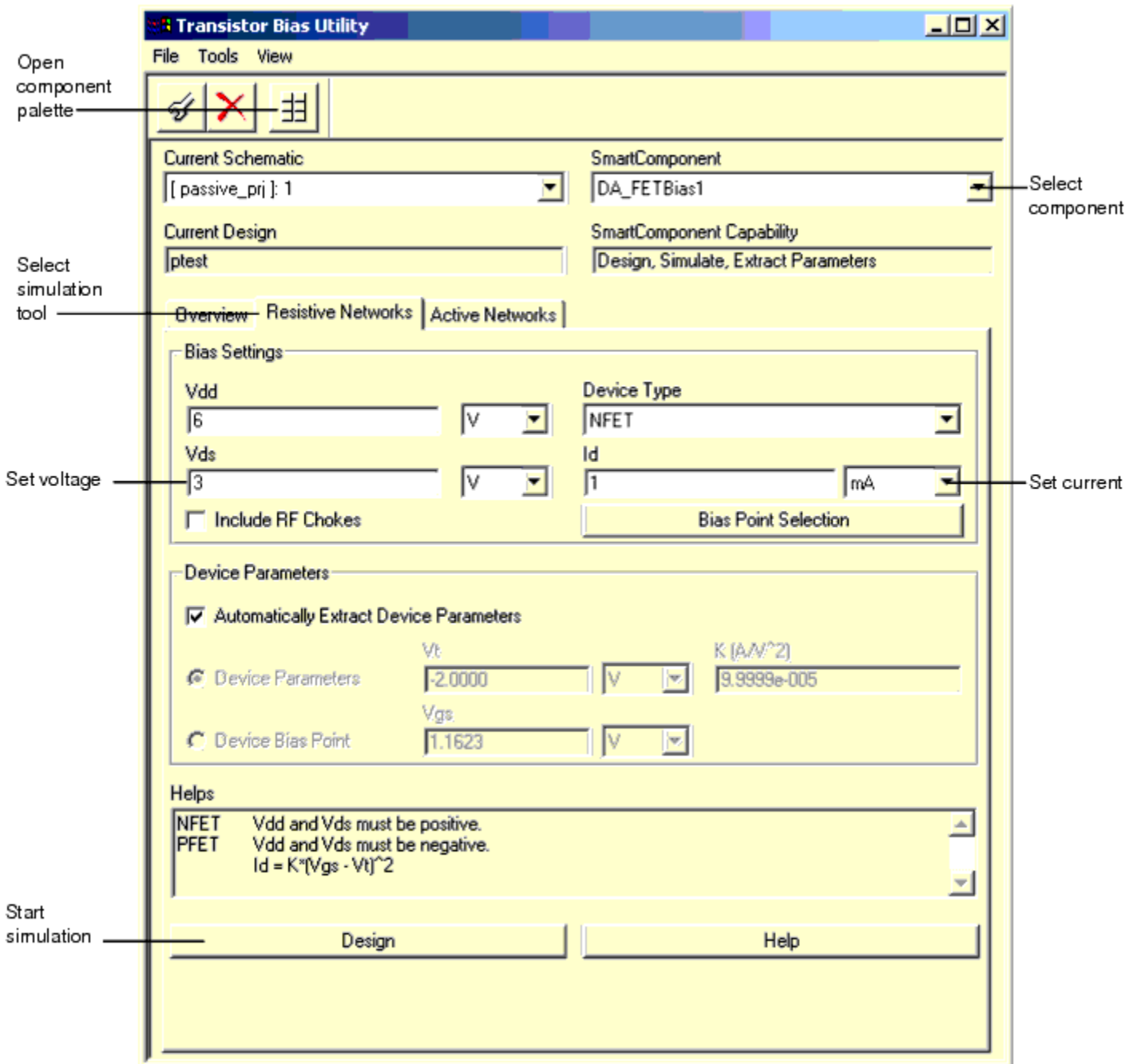
- ✔ Hint  
Expand the list under Tools by clicking the + sign.

2. Select Transistor Bias Utility and click OK to open the tool. The Control window opens.

### Using the Control Window

All Utility features are available from the Control window. The Control window houses menus, a toolbar, and SmartComponent manipulation controls. The menus and toolbar buttons perform the basic functions of design, delete, and display the SmartComponent palette. Full features are available from each of the tab pages on the window. The window can be placed anywhere on the screen. Explore each tab page by clicking on the tab at the top of each page. Explore the window menus as well to familiarize yourself with the basic Utility capabilities.

The pull down lists at the top of the control window are designed to help you navigate multiple schematic windows and SmartComponents. You can use the Current Schematic drop-down list box to select any of the currently opened schematic windows. This field is updated any time Bias Control Window is selected from the DesignGuide menu. From the SmartComponent drop-down list box, you can select any of the SmartComponents on the currently selected schematic window.



To close the Control window:

- Select File > Exit DesignGuide from the Control window menubar. (You can also close the window by clicking the x at the top of the window.)  
Continue the step-by-step example by Designing and Analyzing a Network.

## Designing and Analyzing a Network

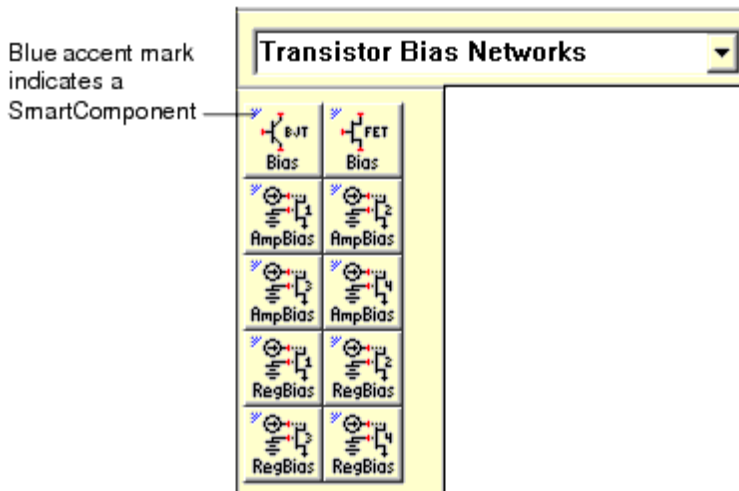
In this step-by-step example, you design and analyze a resistive bias network. A resistive bias network can be designed easily by using the default component parameter settings. Using the Utility follows a normal design flow procedure:

- Select and place a component needed for your design from the component palette ([Displaying the SmartComponent Palette](#) and [Placing Example Component in the Design](#)).
- Provide specifications ([Changing SmartComponent Parameters](#)).
- Design and analyze the component ([Designing the SmartComponent](#)).

**Note**  
Before starting this section of the step-by-step example, confirm your setup ( [Setting Up the Design Environment](#)).

### Displaying the SmartComponent Palette

The DesignGuide contains a SmartComponent palette, Transistor Bias Networks, that provides quick and easy access to the SmartComponents. A blue accent in the upper-left corner of a palette button indicates the component is a SmartComponent.



You can display the SmartComponent palettes in one of these ways:

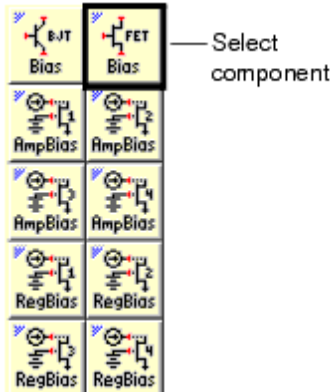
- By clicking Component Palette on the Control window toolbar
- By choosing View > Component Palette from the Control window menu
- By selecting the Transistor Bias palette from the Component Palette drop-down list box in the Schematic window toolbar (directly above the palette).

Continue the example by selecting the Transistor Bias palette. The palette displays in the Schematic window.

### Placing Example Component in the Design

To place a SmartComponent in the design:

1. Click FET Bias on the component palette to select the component.

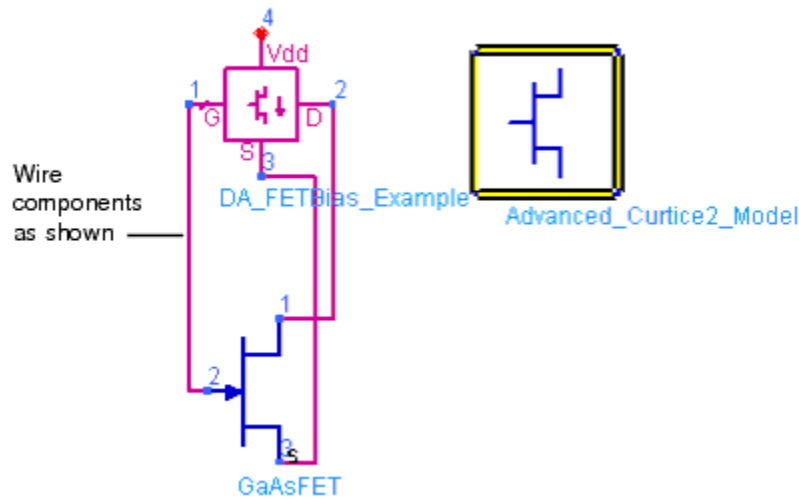


2. Click within the schematic window to place the component.
  - You can change the orientation of the SmartComponent before placement by selecting from the Insert > Component > Component Orientation commands or by selecting Rotate by -90 repeatedly from the schematic toolbar.
  - The place component mode remains active until you choose End Command from the schematic toolbar.

**Note**  
When a SmartComponent is placed initially, a temporary component is used to place and specify the parameters for the SmartComponent. This component does not contain a subnetwork design. After the utility has been used to design the SmartComponent, the temporary component is replaced with a permanent component. The SmartComponent is renamed to DA\_ComponentName\_DesignName and an autogenerated design is placed inside the SmartComponent's subnetwork design file. Subsequently, if the SmartComponent parameters are edited, the utility must be used again to update the subnetwork design file.

Continue the example by placing and wiring the remaining components:

1. Display the Devices-GaAs palette.
2. Place a GaAsFET (GAASN) device and an Advanced Curtice 2 model (AdvCr2) into the design.
3. Wire the gate, drain, and source of the device to the appropriate SmartComponent pins, as shown in Wire the Gate, Drain, and Source in the Example. The Vdd pin does not need to be connected at this time.



Wire the Gate, Drain, and Source in the Example.

## Changing SmartComponent Parameters

Parameters can be changed directly from the DesignGuide Control window.  
To edit the FETBias component parameters:

1. In the Control window, select the FETBias component from the SmartComponent drop-down list. This ensures all changes are referenced to this component.
2. Select the Resistive Networks tab.
3. Set  $V_{ds}$  (drain to source voltage) to 3V and  $I_d$  (drain current) to 1 mA on the control window Bias Settings edit boxes. Leave all other parameters at default.

**Note**  
See [Placing and Editing SmartComponents](#) for details on changing parameters in the design window or component dialog box.

## Designing the SmartComponent

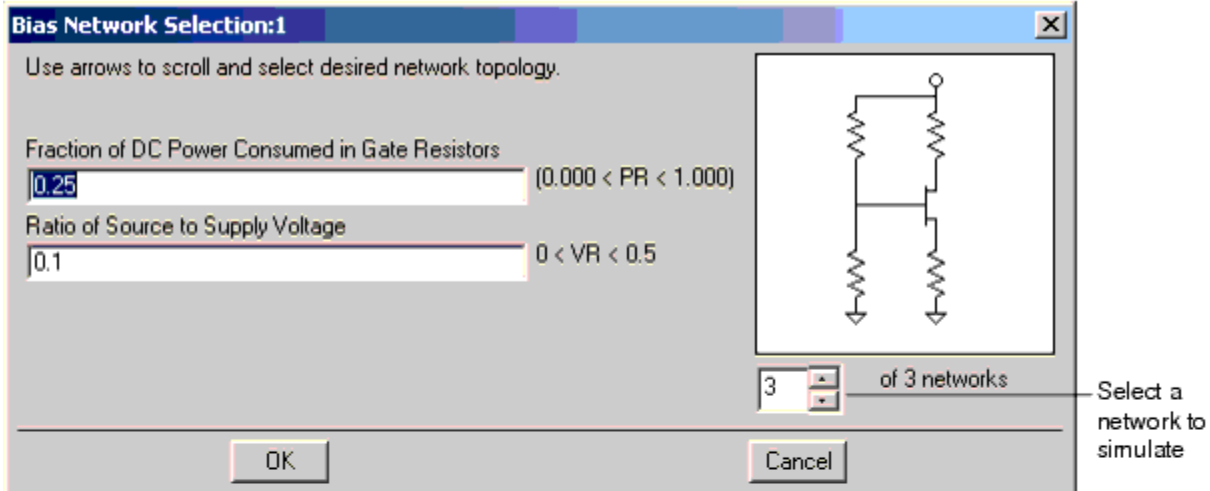
You can design and analyze the SmartComponent from the Control Window.

To start the simulation:

1. On the Resistive Networks tab, click Design to start a simulation. The simulation determines the DC parameters

of the device at the selected bias point. When the simulation has finished, a Bias Network Selection dialog box appears.

2. Select one of the networks and click OK to start a second simulation. (Networks that appear in gray cannot be designed for the current parameter settings.) A second simulation takes place and a data display window summarizing the DC performance of the device appears.



FET Bias Display Assistant Transistor Bias Designer		Need for c
	Vg	Ig
Achieved Bias Voltage/Current	1.76 V	0.00
	Vds	Id
Desired Bias Performance	3.00	1.00
Achieved Bias Performance	3.00	1.00e+0

### Closing the FETBias Analysis Results Window

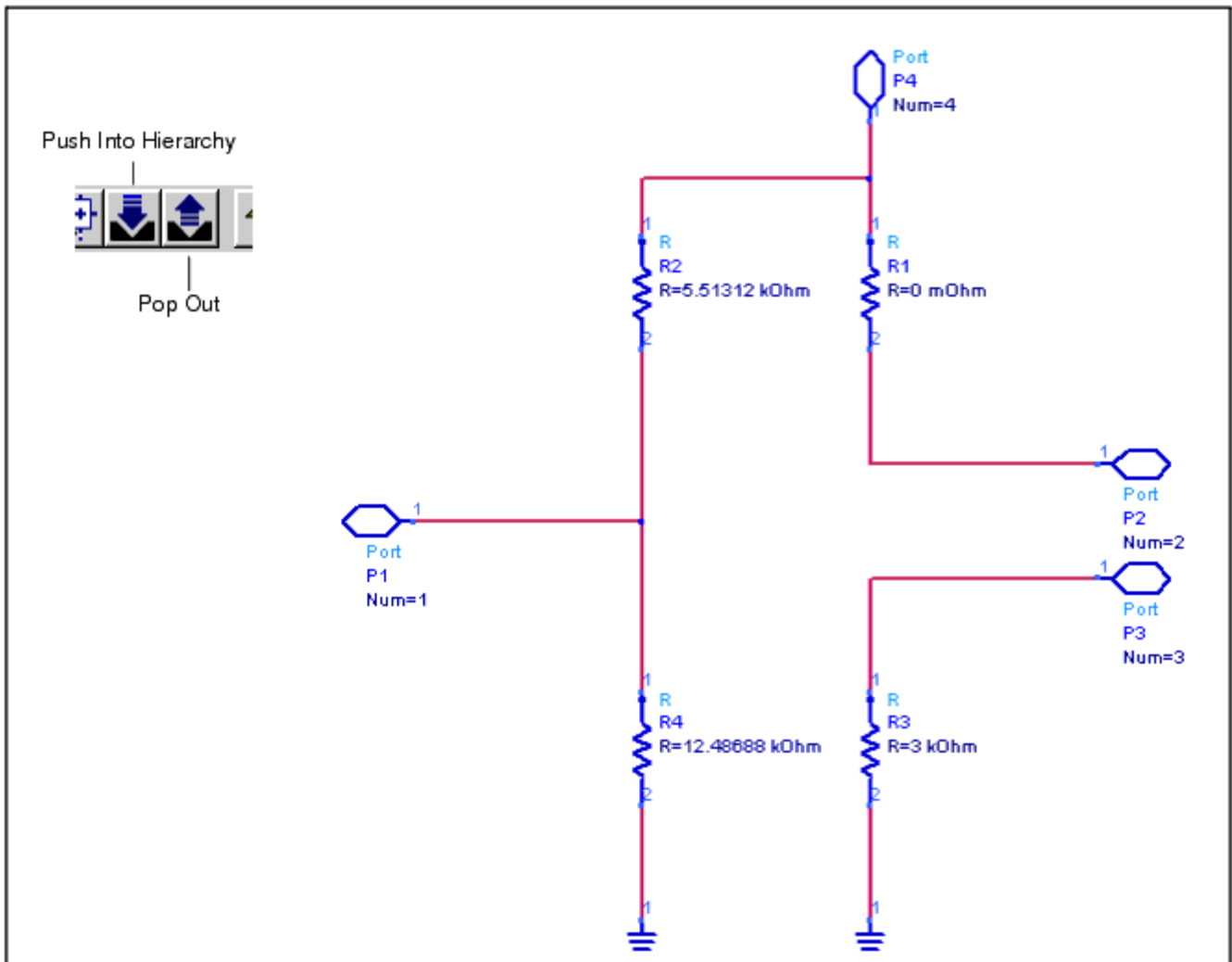
- To close the display window, choose File > Close Window from the menu.

### Examining the Component Design

You can look at the details of the autogenerated design inside the SmartComponent's subnetwork.

To examine the component's subnetwork:

1. Select the component FETBias.
2. Click Push Into Hierarchy on the schematic toolbar.
3. After examining the design, click Pop Out on the schematic toolbar to close the view.



## Deleting the SmartComponent

- To delete the FETBias SmartComponent, choose Tools > Delete SmartComponent from the DesignGuide control window.


**Note**

The Delete button on the DesignGuide control window is different from the Delete button on the ADS schematic window toolbar.

This completes the step-by-step example.

### Using SmartComponents

This Utility provides several SmartComponents representing resistive and active bias networks. SmartComponents are smart sub-network designs that provide the container for specification parameters and a schematic representation of the design when placed into a design. The utility provides automated design and analysis for these SmartComponents. SmartComponents can be placed, copied, edited and deleted like other components in the Advanced Design System. The basics of placement, copying, editing and deleting are described briefly in this section.

 For help with ADS basic features, refer to the [Schematic Capture and Layout](#) documentation.

### Placing and Editing SmartComponents

The components are placed in the schematic by selecting the SmartComponent from the palette and clicking at the point where you want to place the component in the schematic.

### Placing SmartComponents

To place a SmartComponent in the design:

1. In the Schematic window, select the component from the SmartComponent palette.
2. Click within the design window at the location where you want to place the SmartComponent.
  - You can change the orientation of the SmartComponent before placement by selecting from the Insert > Component > Component Orientation commands or by selecting Rotate by -90 repeatedly from the schematic toolbar.
  - The place component mode remains active until you choose End Command from the schematic toolbar.

### Changing Position and Orientation

A SmartComponent is moved by dragging it to any location in the Schematic window.

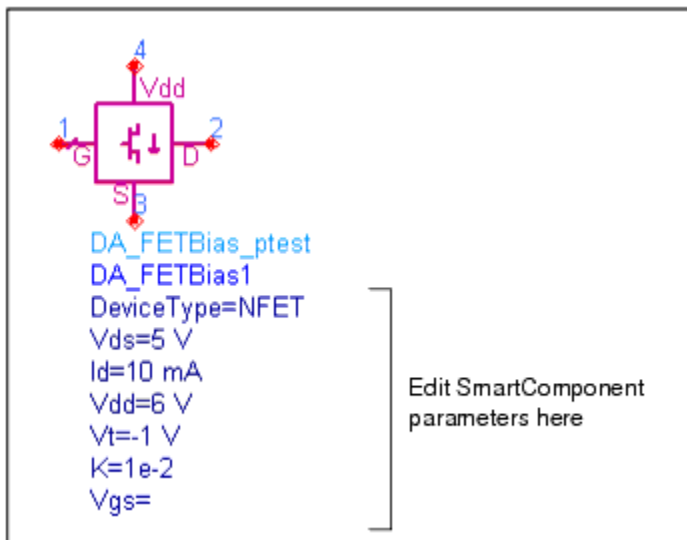
To change the component's orientation:

1. Select Edit > Advanced Rotate > Rotate Around Reference from the Schematic window or select Rotate Items from the toolbar.
2. Click the SmartComponent you want to use.
3. Rotate the component. The rotate mode remains active until you select End Command from the toolbar.

### Editing SmartComponents

Specifications of the SmartComponent are entered directly on the Resistive Networks or Active Networks tab on the Control window. You can also modify the specifications in one of these ways:

- Click the SmartComponent parameters in the schematic window and change them (see The FET Bias Network Component.)
- Double-click the SmartComponent to open a dialog box containing all parameters



The FET Bias Network Component

The SmartComponent design (schematic) can be viewed by pushing into the SmartComponent's subnetwork. See [Examining the Component Design](#).

A SmartComponent subnetwork is empty until the design is generated (see the note in the section [Placing and Editing SmartComponents](#)).

### Copying SmartComponents

SmartComponents can be copied within a design, to another design, or to another Schematic window.

### Copying Within a Design

To copy a SmartComponent to the same design:

1. Click the SmartComponent to be copied.
2. Select Edit > Copy and then Edit > Paste from the schematic window.
3. Click where you want the copy placed.

### Copying Between Designs or Schematic Windows

To copy a SmartComponent to another design:

1. Click the SmartComponent to be copied.
2. Select Edit > Copy from the Schematic window.
3. Display the design or schematic window you want to copy the SmartComponent to.
4. Select Edit > Paste to copy the SmartComponent to the design.
5. Click where you want the component placed.

### Copying a SmartComponent as a Unique Design

Initially, all copied SmartComponents refer to the same SmartComponent design. When the Design Assistant is used to perform a design operation, the Design Assistant transforms each copied SmartComponent into a unique SmartComponent design. A design operation is accomplished from the Utility Control Window.

### Deleting SmartComponents

SmartComponents can be deleted from a design like other components, but completely removing a SmartComponent's files requires the actions described here.

#### Deleting from Current Design

A SmartComponent can be deleted from a design in one of these ways:

- By selecting the component and pressing the Delete key,
- By selecting Delete \*\* from the toolbar,
- By selecting Edit > Delete from the schematic window.



### Note

This procedure does not remove the SmartComponent files from the project directory. To delete files from the project directory, see [Deleting from Current Project](#).

### Deleting from Current Project

To delete a SmartComponent and all associated files from your project:

1. From the DesignGuide Control window, select Tools > Delete SmartComponent or on the toolbar, click Delete SmartComponent.
2. Click the SmartComponent you want deleted. This deletes the SmartComponent from the current design and removes all of its files from your project. The SmartComponent delete mode remains active until you select End Command from the schematic toolbar.

### Deleting Manually Using File System

You can use your computer's file system to delete a SmartComponent by deleting the appropriate files in the network subdirectory of a project. Delete files that start with DA\_ or SA, contain the SmartComponent title, and end with .\_ael, .atf, or .dsn.

## Using SmartComponents as Standalone Components

After SmartComponents are designed and tested, they can be used as standalone components. The Bias Utility is not needed to use them in new designs unless you wish to modify or analyze them. When using the SmartComponent in a design, however, the power supply pins (Vdd, Vcc, Vp, Vm) must be connected to a DC voltage source whose voltage level corresponds the parameter setting.

### Using an Existing SmartComponent Within the Same Project

To use an existing SmartComponent within the same project:

1. Open the Component Library window by selecting Insert > Component > Component Library from the Schematic window or Display Component Library List on the toolbar.
2. Select the project name under All > Sub-networks in the Libraries list at the left of the Component Library window. Available components are listed in the Components list at the right of the Component Library window.

## Advanced Design System 2008

3. Select the SmartComponent in the Components list.
4. Place the SmartComponent into your design by clicking in the Schematic window at the location you wish it placed. The insert mode remains active until you click End Command.

### Using an Existing SmartComponent in Any Project

A library of predesigned reusable SmartComponents can be created by placing the reusable SmartComponents in a project. This project can be included in any project and its SmartComponents can be accessed using the Component Library.

To use an existing SmartComponent in any project:

1. Select File > Include/Remove Projects from the main ADS window.
2. Select the project that contains the SmartComponent from list in the Include & Remove window.
3. Click Include to include the project in the hierarchy and click OK.
4. Open the Component Library window by selecting Insert > Component > Component Library from the Schematic window or Display Component Library List from the toolbar.
5. Select the included project name under All > Sub-networks in the Libraries list at the left of the Component Library window. Available components are listed in the Components list at the right of the Component Library window.
6. Select the SmartComponent in the Components list.
7. Place the SmartComponent into your design by clicking in the Schematic window at the location you where you want to place the component. The insert mode remains active until you click End Command.

### Automated Design and Analysis

The Automated Assistants provide quick design and performance analysis for SmartComponents. Two Automated Assistants are available in this Utility for design:

- Resistive Networks is used to design and simulate the performance of resistive bias networks for BJT and FET devices.
- [Active Networks](#) is used to design and simulate the performance of active bias networks for BJT and FET devices.

Explore each tab page by selecting the associated tab on the control window.

### Resistive Networks

The Resistive Networks tab is used to generate and update the design contained within a resistive bias network SmartComponent from the given specifications. This tool is accessed using the Bias Utility control window. From the control window, full design control is enabled from the Resistive Networks tab. Component design operations can also

be accomplished using the control window menu and toolbar. Any parameter change made from the Resistive Networks tab is reflected on the SmartComponent in the schematic.

Overview **Resistive Networks** Active Networks

**Bias Settings**

Vdd: 6 V

Vds: 3 V

Device Type: NFET

Id: 1 mA

Include RF Chokes

Bias Point Selection

**Device Parameters**

Automatically Extract Device Parameters

Device Parameters

Vt: -2.0000 V

K (A/V<sup>2</sup>): 9.9999e-005

Device Bias Point

Vgs: 1.1623 V

**Helps**

NFET Vdd and Vds must be positive.

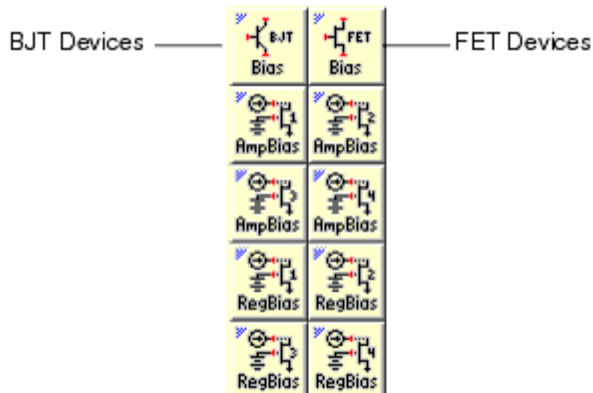
PFET Vdd and Vds must be negative.

$I_d = K \cdot (V_{gs} - V_t)^2$

Design Help

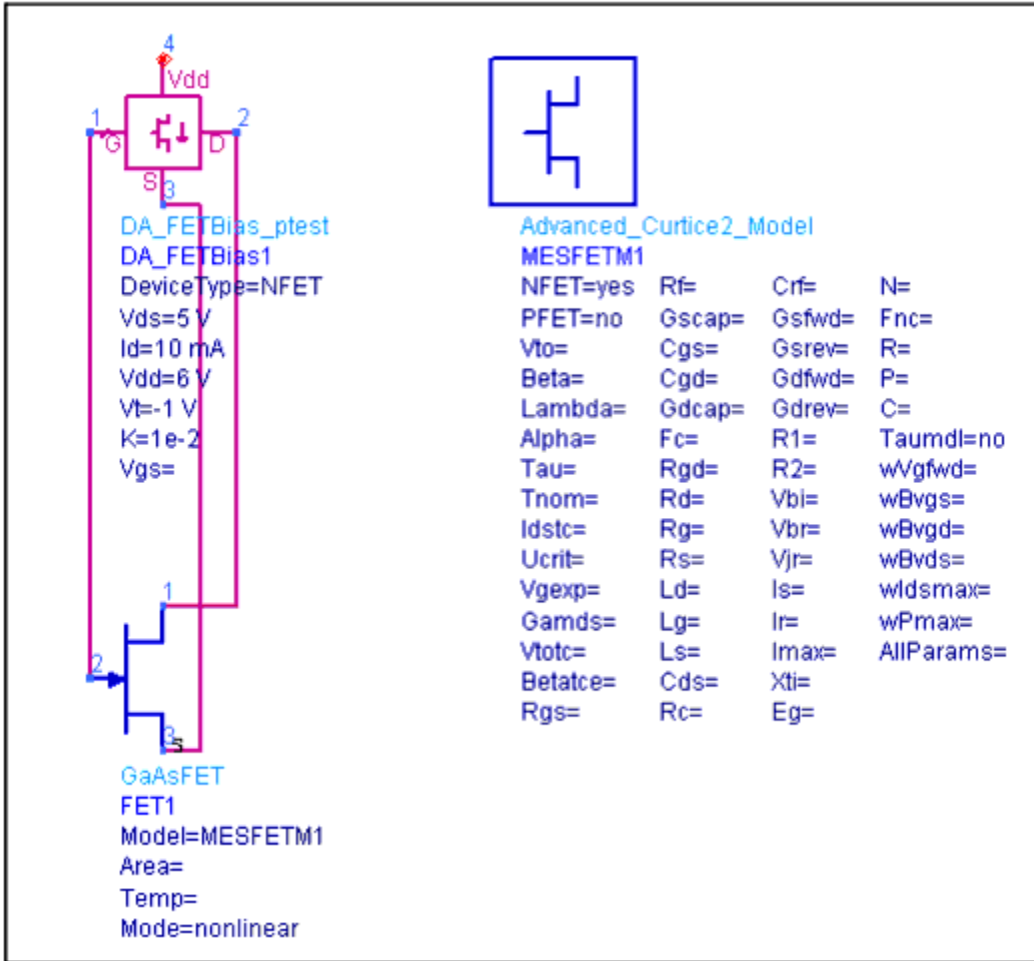
## Resistive Bias Network SmartComponents

Resistive bias networks can be designed for NPN BJT, PNP BJT, NFET, or PFET devices. Two different SmartComponents are available on the Utility palette as shown.



Before designing a network, the SmartComponent pins must be wired to the corresponding pins of the device for which the bias network is to be designed. If the device also has a model associated with it, then this model must be placed on the schematic as well. If the device has pins that should be grounded, this grounding must also be done before a design is attempted.

The SmartComponent supply pin (Vdd or Vcc) does not need to be connected at this time. However, when the SmartComponent is used in a design, this supply pin must be connected to a DC voltage source set at the appropriate supply voltage level. The following image is an example of an appropriate setup in preparation for design.



Appropriate Design Setup for Resistive Bias Network

## BJT Networks

Resistive bias networks for BJT devices have the following options:

Vcc - DC supply voltage value. This is the DC voltage that will be connected to the collected side of the bias network.

Vce - Target bias point collector-to-emitter voltage.

Ic - Target bias point collector current.

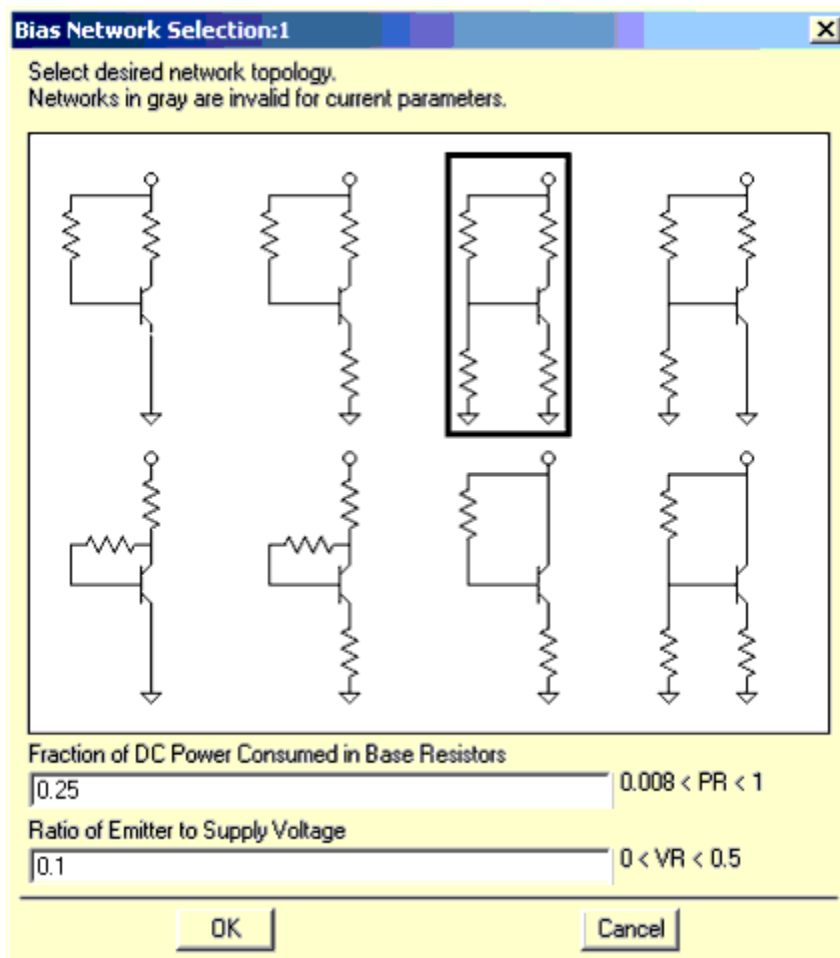
Device Type - BJT device type. For NPN design, all voltages must be positive. For PNP design, all voltages must be negative.

Include RF Chokes - If this option is set, the design incorporates RF choke (DC Feed) elements to isolate the bias network from the RF signal.

Automatically Extract Device Parameters - The DC operation of the device near the operating point is modeled as  $I_c = \beta \times I_b$ , where  $\beta$  is a device parameter and  $I_b$  is the base current at the desired operating point. Furthermore, the device is characterized by a base-emitter voltage drop  $V_{be}$  at the desired operating point. If the option

Automatically Extract Device Parameters is set, the utility attempts to extract beta and Vbe parameters using a simulation. If the target bias point is inappropriate for the device, then this extraction can fail. Alternately, the parameters beta and Vbe can be manually specified.

After parameters have been specified and you have pressed Design, the utility starts the design process. If requested, the device parameters are extracted first. If this extraction is successful, then a dialog opens so you can select the bias network topology. Any networks appearing in gray cannot be designed for the current bias parameters. If all networks are gray, then the bias settings must be altered.



For 3 or 4 resistor topologies, additional specifications must be provided:

**Fraction of DC Power Consumed in Base Resistors** - This parameter sets the current through the base bias network when two resistors are used. The specification is made in terms of the fraction of the total resistive power dissipated in the device that is due to the base resistors.

**Ratio of Emitter to Supply Voltage** - For specifying the emitter resistance, the emitter voltage (specified relative to the supply voltage Vcc) must be specified.

After a suitable network topology has been selected (indicated by a box around the topology), pressing OK completes the design and simulation. A display window opens showing the achieved performance of the network.

### FET Networks

Resistive bias networks for FET devices have the following options:

Vdd - DC supply voltage value. This is the DC voltage that will be connected to the drain side of the bias network.

Vds - Target bias point drain-to-source voltage.

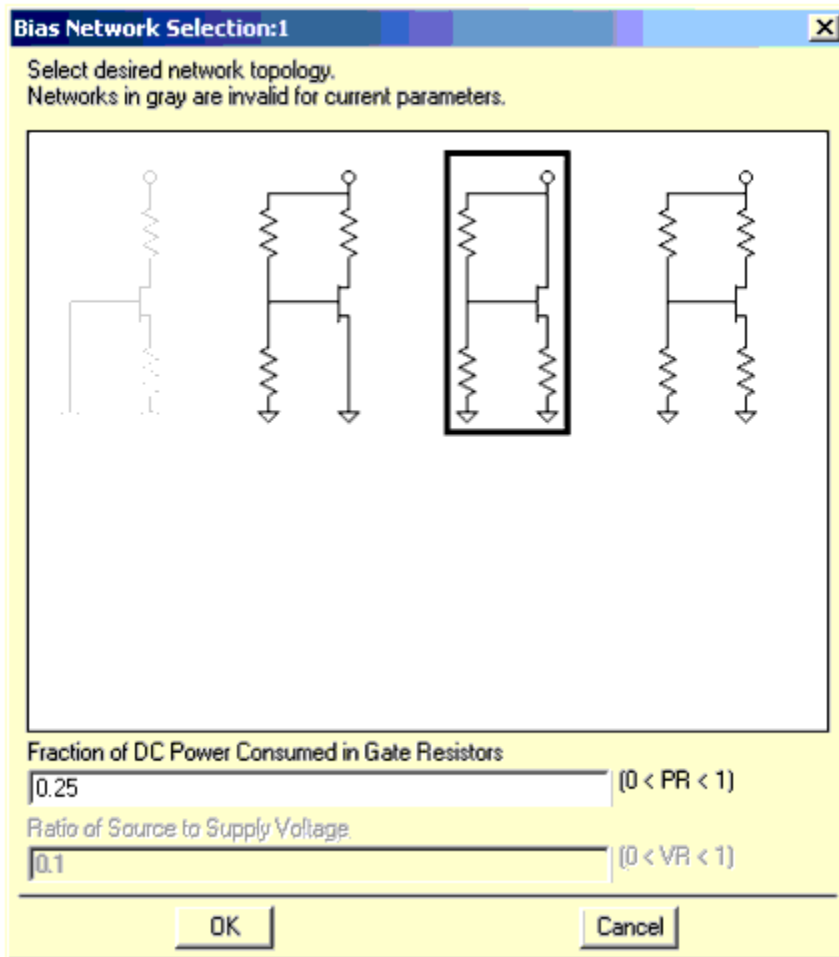
Id - Target bias point drain current.

Device Type - FET device type. For NFET design, Vdd and Vds must be positive. For PFET design, Vdd and Vds must be negative.

Include RF Chokes - If this option is set, the design incorporates RF choke (DC Feed) elements to isolate the bias network from the RF signal.

Automatically Extract Device Parameters - The DC operation of the device near the operating point is modeled as  $I_d = K \times (V_{gs} - V_t)^2$ , where K and Vt are device parameters and Vgs is the gate-to-source voltage at the desired operating point. If the option Automatically Extract Device Parameters is set, the utility attempts to extract these parameters using a simulation. If the target bias point is inappropriate for the device, then this extraction can fail. Alternately, either the parameters K and Vt or the bias point Vgs can be manually specified.

After parameters have been specified and you have pressed Design , the utility starts the design process. If requested, the device parameters are extracted first. If this extraction is successful, then a dialog opens so you can select the bias network topology. Any networks appearing in gray cannot be designed for the current bias parameters. If all networks are gray, then the bias settings must be altered.



For 3 or 4 resistor topologies, additional specifications must be provided:

**Fraction of DC Power Consumed in Gate Resistors** - This parameter sets the current through the gate bias network when two resistors are used. The specification is made in terms of the fraction of the total resistive power dissipated in the device that is due to the gate resistors.

**Ratio of Source to Supply Voltage** - For specifying the source resistance, the source voltage (specified relative to the supply voltage  $V_{dd}$ ) must be specified.

After a suitable network topology has been selected (indicated by a box around the topology), pressing OK completes the design and simulation. A display window opens showing the achieved performance of the network.

### Bias Point Selection for Resistive Bias Networks

Typically, selection of the bias point is performed based upon specifications provided by device manufacturers. To assist in this selection process, simulation and display templates are provided. You can use these templates to choose the bias point based upon optimal Class A operation for power amplifiers, or to achieve target gain or noise figure specifications for small-signal amplifiers. The templates contain text on the schematic and display windows indicating

the sequence of steps to follow to assess the device performance.

After a device bias point has been determined from these templates, the schematic template must be closed and the design containing the original SmartComponent must be visible before the design can proceed.

## Active Networks

The Active Networks tab is used to generate and update the design contained within an active bias network SmartComponent from the given specifications. This tool is accessed using the Bias Utility control window. From the control window, full design control is enabled from the Active Networks tab. Component design operations can also be accomplished using the control window menu and toolbar. Any parameter change made from the Active Networks tab is reflected on the SmartComponent in the schematic.

Overview | Resistive Networks | **Active Networks**

**Supply Settings**

Positive Supply (Vp)  V   
 Negative Supply (Vm)  V

**Bias Settings**

Device Voltage (VBias)  V   Include RF Chokes

Device 1 Current (I1)  mA   
 Device 2 Current (I2)

Device 3 Current (I3)    
 Device 4 Current (I4)

**Bias Point Selection**

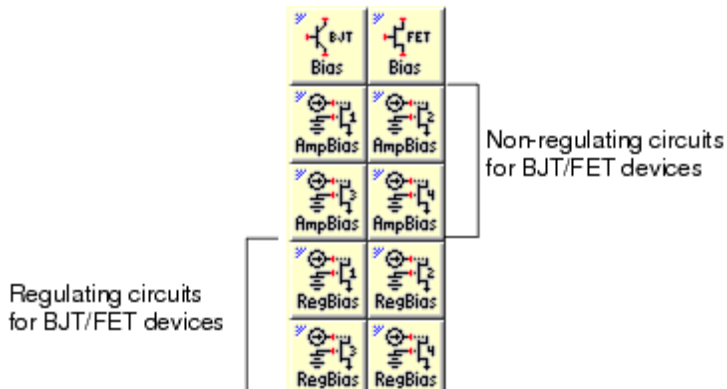
FET  BJT  Device Number

**Helps**

The device can be a grounded source NFET or a grounded emitter NPN BJT.

## Active Bias Network SmartComponents

Active bias networks can be designed for NPN BJT or NFET devices. Eight different SmartComponents are available on the Utility palette as shown.



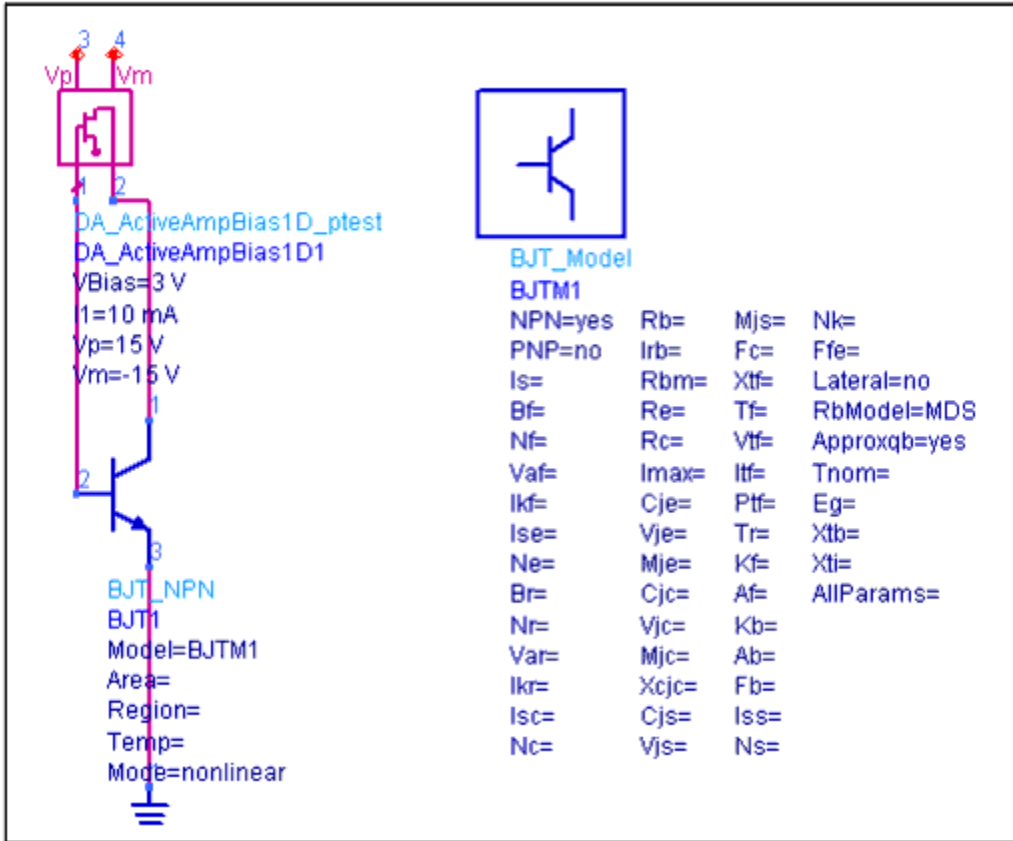
Active bias networks use operational amplifiers to create a network that can offer the specified bias point independent of the device characteristics. The bias voltage and current are set independently. A non-regulating (OpAmp based) design provides a simple network with low parts count. However, the performance can vary as a function of the tolerances of the parts used to fabricate the network. The regulating designs use a zener diode to provide a more tolerant design at the expense of a more complicated network.

Before designing a network, the SmartComponent pins must be wired to the corresponding pins of the device for which the bias network is to be designed. If the device also has a model associated with it, then this model must be placed on the schematic as well. If the device has pins that should be grounded, then this grounding must also be done before a design is attempted. Since these bias networks can be used only for grounded source FET or grounded emitter BJT devices, these pins on the device must be grounded.

The SmartComponent supply pins ( $V_p$  and  $V_m$ ) do not need to be connected at this time. However, when the SmartComponent is used in a design, these supply pins must be connected to DC voltage sources set at the appropriate supply voltage levels.

For each type of network (non-regulating and regulating), the four SmartComponents available can accommodate different numbers of devices (1 through 4 devices). All devices biased by a given SmartComponent must share the same bias voltage ( $V_{ce}$  or  $V_{ds}$ ), but can have independent bias currents.

Appropriate Design Setup for Active Bias Networks is an example of an appropriate setup in preparation for design.



Appropriate Design Setup for Active Bias Networks

## Network Design

Active bias networks for NPN BJT and NFET devices have the following options:

Positive Supply (Vp) - DC positive supply voltage value. This DC voltage runs the operational amplifiers that are used to create the bias.

Negative Supply (Vm) - DC negative supply voltage value. This DC voltage runs the operational amplifiers that are used to create the bias.

Device Voltage (VBias) - Target bias point collector-to-emitter or drain-to-source voltage for all devices.

Device Current (I) - Target bias point collector or drain current. Each device can have a unique bias current.

Include RF Chokes - If this option is set, the design incorporates RF choke (DC Feed) elements to isolate the bias network from the RF signal.

After parameters have been specified and you have pressed Design, the utility starts the design process. A simulation is performed after the design is complete, and a display window opens showing the achieved performance.

### Regulating Networks

The Regulating Bias Networks (RegBias) use a zener diode to regulate the actual bias voltage level achieved. This makes for a design that is more tolerant to variations in component values. Furthermore, these networks use RC networks such that the drain voltage is applied before the gate voltage is applied. For many FET devices, this dramatically reduces device failure due to damaged gate oxide.

For these networks, the zener diode voltage (  $V_{zener}$  ) and maximum power rating of the resistors used in the network (  $P_{max}$  ) must be specified. These parameters cannot be specified on the control window, and therefore must be specified directly on the SmartComponent on the schematic window or using the parameter dialog box that appears by double-clicking on the SmartComponent.

### Bias Point Selection for Active Bias Networks

Typically, selection of the bias point is performed based upon specifications provided by device manufacturers. To assist in this selection process, simulation and display templates are provided. You can use these templates to choose the bias point based upon optimal Class A operation for power amplifiers, or to achieve target gain or noise figure specifications for small-signal amplifiers. The templates contain text on the schematic and display windows indicating the sequence of steps to follow to assess the device performance.

To use this capability for Active Bias networks, the number of the device (1-4) as well as the device type (BJT or FET) connected to the SmartComponent must be specified. Pressing Open Selection Template opens the appropriate schematic and display templates.

After a device bias point has been determined from these templates, the schematic template must be closed and the design containing the original SmartComponent must be visible before the design can proceed.

## Load Pull Measurement Data Import Utility

The Load Pull Measurement Data Import utility is used to import Maury or Focus Loadpull measured loadpull data in Agilent's Advanced Design System (ADS) for interpolation and generation of load and source pull contours. It is assumed that the data is generated for a constant bias, fixed frequency and constant input power.

The Load Pull Measurement Data Import utility can be selected and installed during installation process of ADS. After installation, the Load Pull utility can be accessed from Schematic window.

The process includes:

- [Importing Loadpull Measurement Data](#)

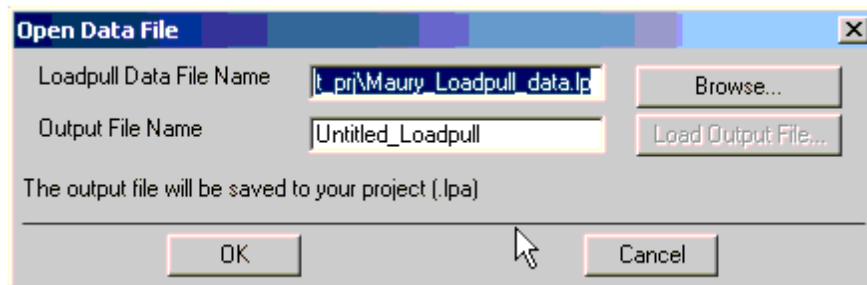
- [Preparing Data for Interpolation](#)
- [Interpolating the Data and Displaying Results](#)

### Importing Loadpull Measurement Data

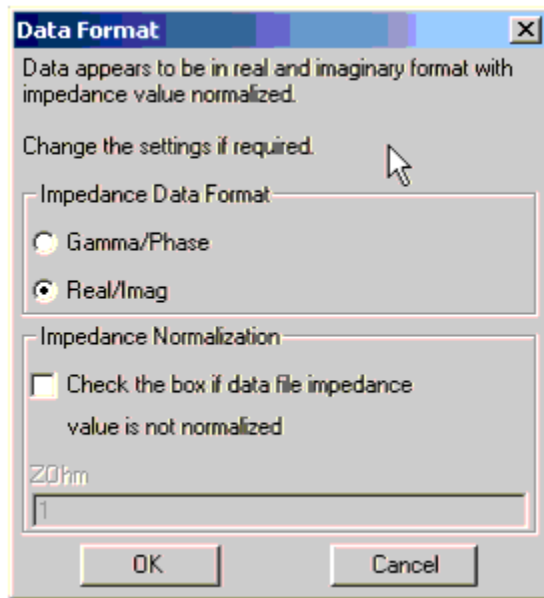
Before you can use Loadpull Measurement Data Import utility, you must have measured loadpull data available on your system. If you do not have measured loadpull data, you can access and copy sample measured data file for Maury and Focus microwave system to your project directory by selecting DesignGuides > Loadpull > Sample Loadpull Data File.

To import loadpull measurement data:

1. In the Schematic window, choose DesignGuides > Loadpull > Loadpull Measurement Data Import Utility. Click OK. A dialog box appears.
2. Specify the File Format option. You can import either Maury or Focus measured data.
3. Select Select Loadpull Data File to open the File Selection dialog box. You can select either Browse or Load Output File.
  - Use the Browse option to select a specific data file from the project directory and define the output file name.
  - Use the Load Output File option to load a previously-prepared load pull data file. You can use this option if you have successfully prepared loadpull data earlier and you want to load it now. All the prepared loadpull file have an extension .lp . When selecting the data file, all the Measurement Variable for contour plots loads directly. You can add a Measurement Variable to Trace and Generate Contours.
4. Select the measured data file using Browse button and define the Output File Name. Click OK.



During selection of loadpull-measured data file, the format of the loadpull impedance data is determined. If the impedance value is normalized to its characteristic impedance, the Data Format dialog box displayed during this process reflects these settings.



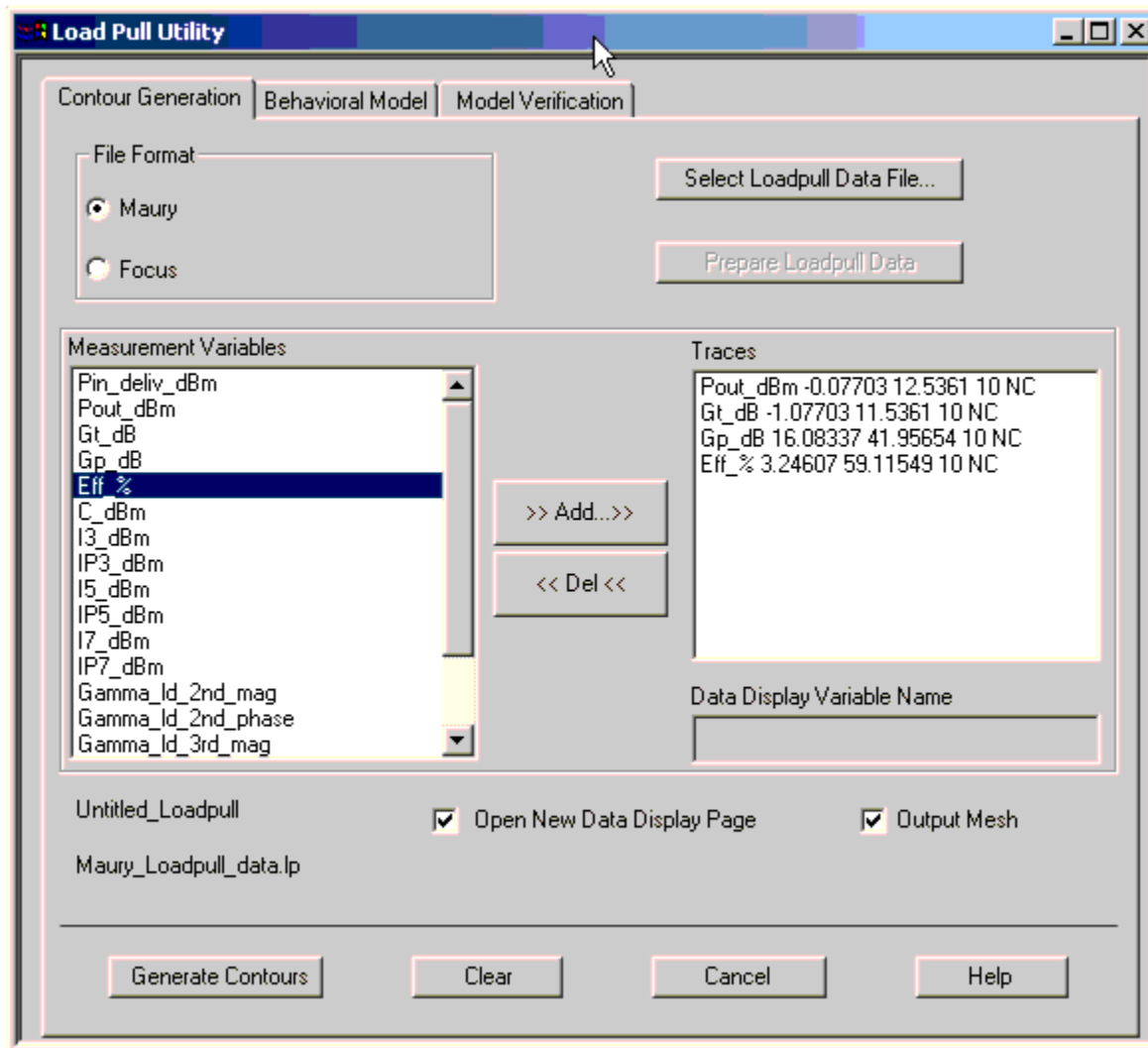
In some cases, it is difficult to determine the data format correctly due to non-availability of measured data points across the entire Smith Chart region. In this case, you must make the changes manually. If the impedance values are not normalized and appear in the load pull file as, for instance,  $52+j \times 32$ , check the Impedance Normalization check box and set the value of  $Z_0\text{hm}$  to the applicable normalized impedance (such as, 50 ohms).

## Preparing Data for Interpolation

In this step, the tool reads the measurement points and prepares the data for interpolation. You also have an opportunity to select the measurement variables you want to plot.

To prepare data for interpolation:

1. In the Load Pull Utility dialog, select Prepare Loadpull Data to prepare the data for interpolation. The process takes all the measurement points and generates a triangular mesh to perform linear interpolation on scattered data. After the data is prepared for interpolation, the measurement variables for which the contours can be generated appear in the Measurement Variable field. The utility does not restrict the number of variables you can plot simultaneously. You can generate loadpull contours for multiple variables at the same time, provided you have valid measurement data to generate loadpull contours.
2. Select the variable from the Measurement Variable field and add to the Trace field. During this process you enter the minimum and maximum bounds for your contour plots and the number of contours you want to generate within defined bounds. Add all the Measurement Variables you want to plot to the Trace field.



3. Select Open New Data Display Page so you can plot loadpull contours on Smith Chart or on Polar Chart. (Note: to view contours on a rectangular grid, select Plot real(Variable\_Name) vs. imag(Variable\_Name)). For details on using the data display, refer to [Plots and Lists](#) in the Data Display documentation. Select the Output Mesh option to export the triangular mesh generated during Prepare Loadpull Data to a data display page. The output mesh can be plotted by plotting the variable Zmesh on the Smith Chart or Polar Chart. You can use the output mesh to view the measurement points and for debugging purposes.

## Interpolating the Data and Displaying Results

The interpolation process generates load pull contours and displays the results.

To interpolate the data:

1. Click Generate Contours to start the interpolation process. This process interpolates the prepared data and

## Advanced Design System 2008

generates load pull contours. At the end of the interpolation process, the data display opens.

2. Select the type of display from the list. During the translation process, the measurement variable name changes to remove all special character and a prefix "Z" is appended at the beginning. To view the new assigned name in the Data Display Variable Name, select the name in the Trace window.

