



Agilent Technologies

Filter DesignGuide

August 2005

Notice

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

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

Warranty

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

Restricted Rights Legend

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

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

Acknowledgments

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

Microsoft[®], 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.

Contents

1 Introducing the Filter Design Guide

Getting Started	1-1
Setting Preferences	1-1
Filter Design Flow	1-3
SmartComponents	1-4
Automated Assistants	1-5
Accessing the DesignGuide	1-6
The Filter DesignGuide Control Window	1-8
SmartComponent Palette Access	1-9
Using the Filter Design Guide.....	1-11
Create a New Design	1-11
Open the DesignGuide Control Window.....	1-11
Design and Analyze a Lowpass Filter	1-11
Analyze Yield of the Lowpass Filter.....	1-13

2 Using SmartComponents

Overview.....	2-1
Placing SmartComponents.....	2-1
Copying SmartComponents	2-2
Copying Within A Design.....	2-2
Copying Between Designs or Schematic Windows	2-2
Editing SmartComponents	2-3
Position and Orientation	2-3
Parameters	2-3
Deleting SmartComponents	2-3
Delete From Current Design.....	2-4
Delete From Current Project.....	2-4
Delete Manually Using File System.....	2-4
Design, Analysis, and Yield	2-4
Stand-Alone SmartComponent Usage	2-5
Using an Existing SmartComponent Within the Same Project.....	2-5

3 Using Automated Assistants

Filter Assistant	3-1
Impedances	3-2
Order	3-2
Design Information	3-2
Response Type.....	3-2
Frequency.....	3-2
Attenuation	3-3

Realizations	3-3
Design	3-3
Simulation Assistant	3-4
Simulation Frequency Sweep	3-5
Automatically Display Results.....	3-5
Starting the Simulation	3-6
Simulation Templates	3-6
Yield Assistant	3-6
Simulation Frequency Sweep	3-7
Statistical Components.....	3-7
Yield Optimization.....	3-9
Yield/Optimization Goals	3-9
Automatically Display Results.....	3-10
Starting the Simulation	3-10
Yield Results.....	3-10
Yield Templates	3-10
Display Assistant	3-11
Display Template Features	3-11
Basic Layout	3-12
Typical Area One Graph	3-12
Typical Area Two Graphs.....	3-13
Typical Area Three Templates	3-14
Display Assistant Operation	3-14
Opening a Display	3-14
Display Templates	3-15
Transformation Assistant	3-15
Accessing the Transformation Assistant.....	3-15
Selecting a Transformation Type	3-16
Component Selection	3-18
Transformation Buttons.....	3-18
Changing Component Type.....	3-18
Transmission Line Types	3-19
Additional Transformation Functions	3-19

4 SmartComponent Reference

SmartComponent List.....	4-1
LCLowpassDT (Doubly Terminated Lowpass Filter).....	4-2
LCHighpassDT (Doubly Terminated Highpass Filter).....	4-4
LCBandpassDT (Doubly Terminated Bandpass Filter).....	4-6
LCBandstopDT (Doubly Terminated Bandstop Filter)	4-8
LCLowpassST (Singly Terminated Lowpass Filter)	4-10
LCHighpassST (Singly Terminated Highpass Filter)	4-12

LCBandpassST (Singly Terminated Bandpass Filter)	4-14
LCBandstopST (Singly Terminated Bandstop Filter)	4-16

Index

Chapter 1: Introducing the Filter Design Guide

This chapter introduces the Filter DesignGuide. This manual assumes you have installed the DesignGuide with appropriate licensing codewords.

Getting Started

The Filter DesignGuide provides SmartComponents and automated-assistants for the design, simulation, and performance analysis of common passive filters. Automated-assistants include a Filter Assistant, Simulation Assistant, Yield Assistant and Display Assistant, which enable you to quickly create and verify a design. All SmartComponents can be modified when selected. You simply select a SmartComponent and with little effort redesign or verify their performance.

The complexity of the Advanced Design System (ADS) is made easily accessible through the automated-assistants. This enables a first-time or casual ADS user to begin using the capability of ADS quickly, and experienced ADS users will be able to perform tasks faster than ever before. As an example, a singly terminated bandstop elliptical filter can be designed, verified and a layout generated in a few minutes saving the designer substantial time.

Note This manual is written describing and showing access through the cascading menu preference. If you are running the program through the selection dialog box method, the appearance and interface will be slightly different.

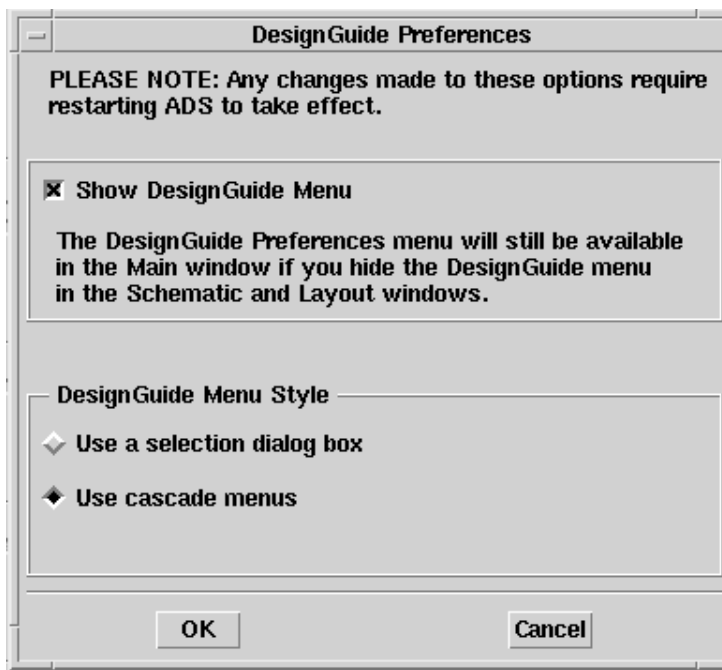
Setting Preferences

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.

Note This manual is written describing and showing access through the cascading menu preference. If you are running the program through the selection dialog box method, the appearance and interface will be slightly different.

To configure access through menus or dialog boxes:

1. From the Main or Schematic window, select **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. Thus the dialog box menu style might be best for these situations.

The ADS Main window DesignGuide menu contains the following choices:

DesignGuide Developer Studio > Start DesignGuide Studio is only available on this menu if you have installed the DesignGuide Developer Studio. It launches 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. It brings up 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 brings up 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 brings up a list of your installed DesignGuides. Select any that you would like to uninstall and choose the *Remove* button.

Preferences brings up 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 will be removed if this box is unchecked.
- Select your preferred interface method (cascading menus vs. dialog boxes).

Filter Design Flow

The Filter DesignGuide follows standard design procedure:

1. Select a component needed for your design.
2. Provide specifications.
3. Design and analyze the component.
4. If the component performance needs adjustment, optimize the component.

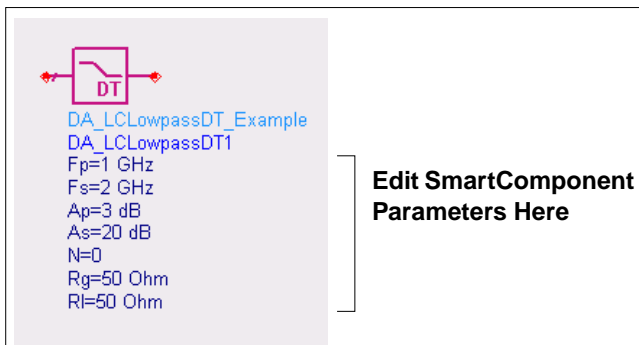
There are two important general concepts: SmartComponents and Automated Assistants.

SmartComponents

The DesignGuide provides many passive SmartComponents such as doubly and singly terminated filters. SmartComponents contain specification parameters and a schematic representation of the design.

SmartComponents are manipulated using several Automated Assistants. These assistants enable you to easily design and simulate the SmartComponents.

SmartComponents are smart sub-network designs that can be placed into a schematic. The Doubly Terminated Lowpass Filter Component is shown here.



The components are placed in the schematic by selecting the desired SmartComponent from the palette and clicking at the point where you want them placed in the schematic. The desired specifications of the SmartComponent are entered by clicking on its parameters and changing them. In addition, a dialog box containing all parameters is available by double-clicking on the SmartComponent.

The SmartComponent design (schematic) can be viewed by pushing into the SmartComponent's subnetwork. A SmartComponent subnetwork is empty until the Design Assistant is used to generate the design. For details on the SmartComponents, refer to [Chapter 4, SmartComponent Reference](#).

Hint Place a doubly terminated lowpass filter SmartComponent into a schematic by clicking the *LowpassDT* palette button and clicking within the Schematic window at the desired placement location.

Creating a Design using SmartComponents

1. Choose and place a SmartComponent.
2. Edit the SmartComponent parameters (specifications).
3. Design the SmartComponent using the Filter Assistant.
4. Analyze the SmartComponent's performance using the Simulation and Yield Assistants.
5. Display the performance of the SmartComponent using the Display Assistant.

Automated Assistants

Five Automated Assistants are available in this DesignGuide. They provide quick design, simulation, yield analysis, and performance display for SmartComponents. They also enable transformation of lumped elements to transmission line elements. Each Automated Assistant has a command page that is accessed from DesignGuide control window. Explore each command page by selecting the associated tab (or dialog) on the control window. Following are descriptions of each Automated Assistant.



Filter Assistant is used to generate/update a SmartComponent's schematic design for filter networks. After a SmartComponent is placed and the parameters are specified, you start the Filter Assistant to design the component. Subsequently, if the parameters of the SmartComponent are modified, you start the Filter Assistant again to update the design. For more information, refer to ["Filter Assistant" on page 3-1](#).



Simulation Assistant is used to automatically perform a simulation of a SmartComponent. After a SmartComponent has been designed using the Filter Assistant, you start the Simulation Assistant to automatically analyze the component. You can easily examine the simulation results using the Display Assistant. For more information, refer to ["Simulation Assistant" on page 3-4](#).



Yield Assistant is used to automatically perform a simulation of a SmartComponent's network sensitivity. After a SmartComponent has been designed using the Filter Assistant, you start the Yield Assistant to automatically analyze the component. You can easily examine the sensitivity results using the Display Assistant. For more information, refer to ["Yield Assistant" on page 3-6](#).



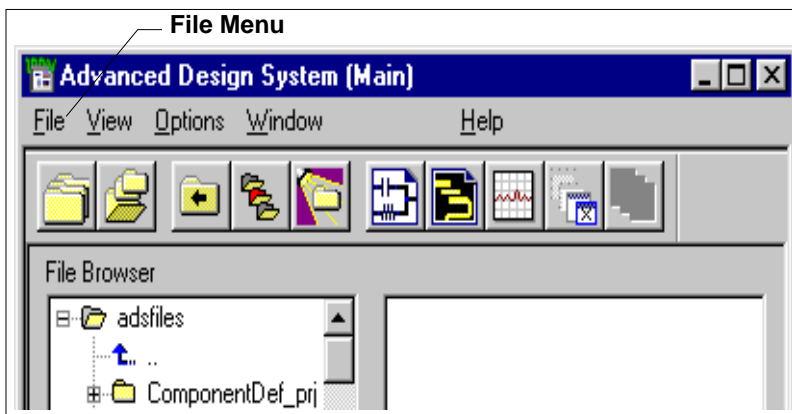
Display Assistant is used to automatically display the analysis results generated using the Simulation Assistant. By starting the Display Assistant, you can quickly display the results generated from the most recent simulation of a SmartComponent. For more information, refer to [“Display Assistant” on page 3-11](#).



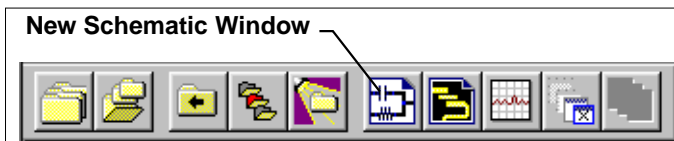
Transformation Assistant provides an interactive environment for transforming lumped elements to transmission lines within a designed SmartComponent. For more information, refer to [“Transformation Assistant” on page 3-15](#).

Accessing the DesignGuide

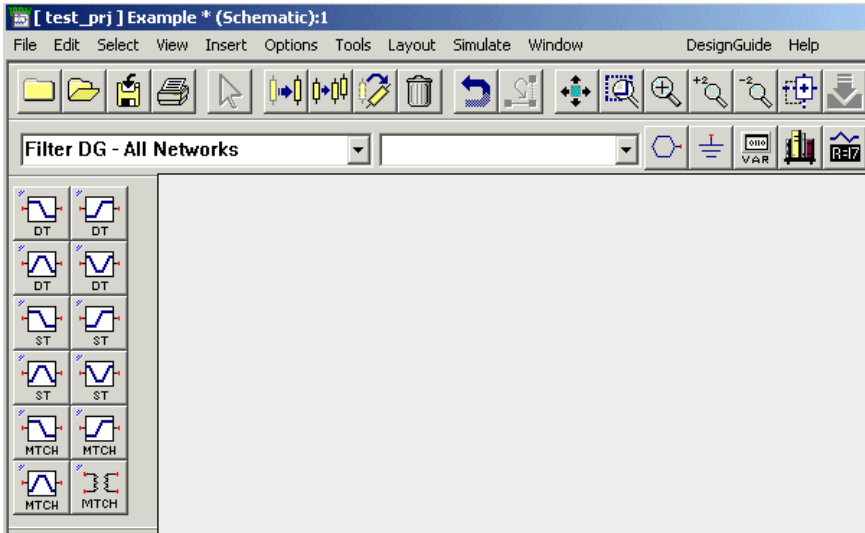
The Filter DesignGuide is accessed from a Schematic window within ADS. First you must create or open a project. From the ADS main window, select *File > New Project* or *File > Open Project*, as shown here. For this example, create a new project called *QuickStart*.



To open a Schematic window, select *Window > New Schematic* or click the *New Schematic Window* toolbar button.



A new Schematic window appears, as shown here. The DesignGuide features are accessed using the menu, control window, and SmartComponent palettes. The Filter DesignGuide menu is available from the DesignGuide menu.

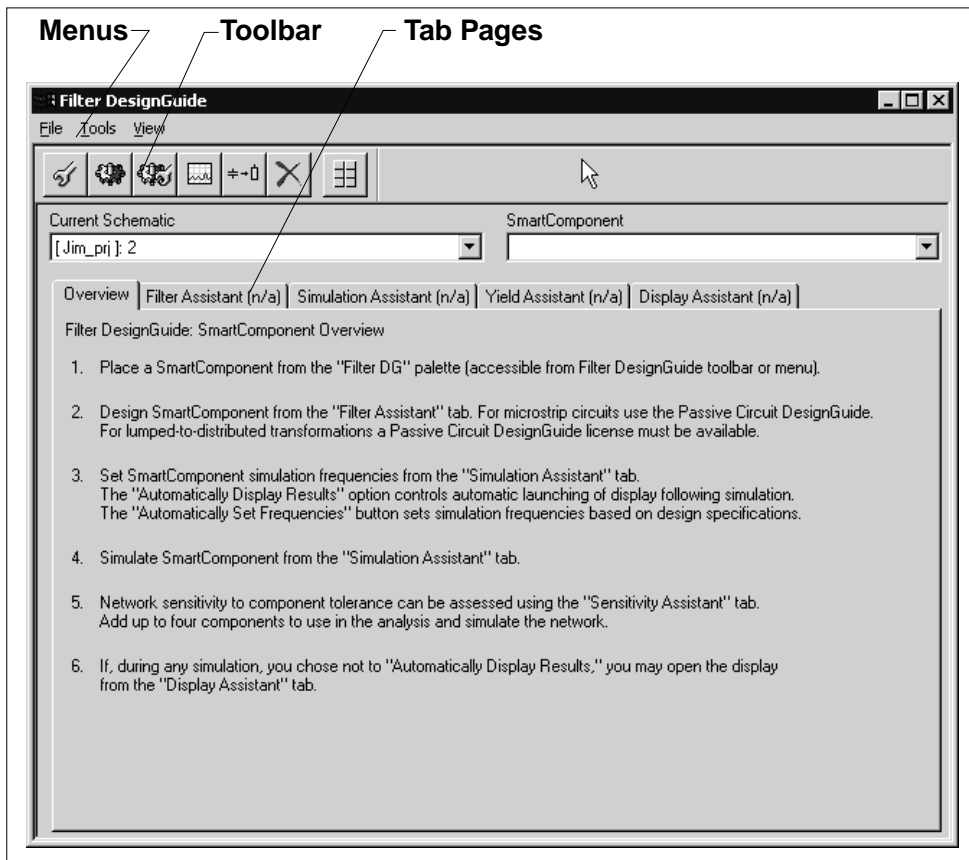


Note Depending on how your ADS preferences are set, a Schematic window may automatically appear when you create or open a project.

The Filter DesignGuide Control Window

All DesignGuide features are available from the Control Window that houses menus, a toolbar, and SmartComponent manipulation controls.

Filter DesignGuide Control Window



The window can be placed anywhere on the screen. To access the Control Window, select the *Filter Control Window* sub-menu from the *Filter* menu. The menus and toolbar buttons perform the basic functions for each automated Assistant (Design, Simulate, Yield, and Display) as well as display the SmartComponent palettes. Full features are available from each of the tab pages on the window. Explore each Automated Assistant tab page by clicking on the tab at the top of each page. Explore

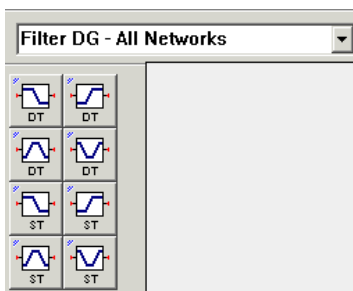
the window menus as well to familiarize yourself with the basic DesignGuide capabilities.

The window includes the following features and functions:

- You can place the window anywhere on the screen:
- With the fields at the top of the Control Window, you can navigate multiple Schematic windows and SmartComponents.
 - The *Current Schematic* drop-down list box enables you to select any of the currently opened Schematic windows. This field is also updated when *Passive Circuit Control Window* is selected from the *Passive Circuit* menu.
 - The current design name is also displayed below the *Current Schematic*. The *SmartComponent* drop-down list box enables you to select any of the SmartComponents on the currently selected schematic window.
 - The *SmartComponent Capability* field informs you of what functions (design, simulate, optimize, and display) the DesignGuide can perform for that particular component.
- To close the Control Window, select *File > Exit DesignGuide* from the Control Window menu bar. The window may also be closed using the window close feature of the operating system (a button marked with an 'x' at the top of the window).

SmartComponent Palette Access

The SmartComponent palettes are displayed by using the right button group in the control window toolbar or from the control window menu *View* menu. (They can also be chosen from the palette list box in the Schematic window toolbar.) The *Filter DG - All Networks* palette contains all of the filter SmartComponents. A blue accent in the upper-left corner of a palette button indicates the component is a SmartComponent.



Using the Filter Design Guide

This step-by-step example will take you through the design, analysis and sensitivity simulation of a doubly terminated lowpass Chebyshev filter. After completing this example, you should have a basic understanding of the DesignGuide.

Note This example is based on the assumption that you are familiar with the basic features of Advanced Design System.

Create a New Design

A new schematic design is needed to contain the lowpass component for the following exercises. Follow these simple steps to create a new design named Example.

1. Open a new Schematic window.
2. Select **File > New Design** from the Schematic window to create a Analog/RF Network design named Example.

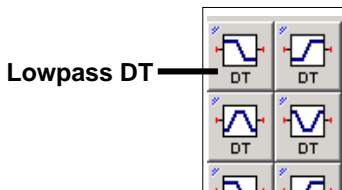
Open the DesignGuide Control Window

1. From the DesignGuide menu on the ADS Schematic window, select **Filter**.
2. From the *Filter* window, select **Filter Control Window** and click **OK**.

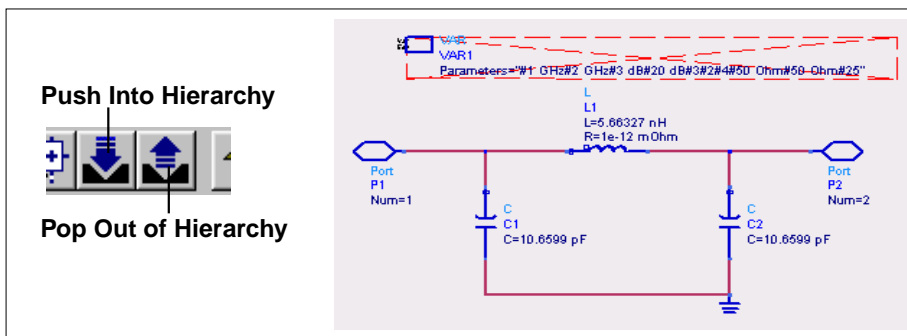
Design and Analyze a Lowpass Filter

A lowpass filter can easily be designed using the default component parameter settings. Follow these simple steps to design a lowpass filter.

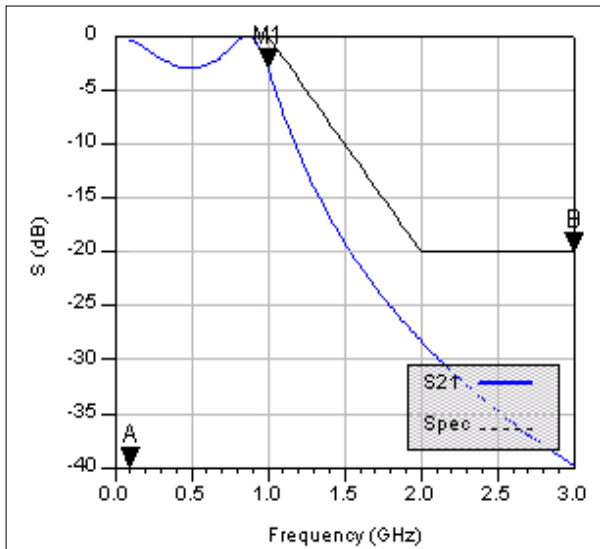
1. **Display the Filter DG - All Networks palette.** (Refer to the section, [“SmartComponent Palette Access” on page 1-9.](#))
2. **Insert a LowpassDT component in the schematic.** Click LowpassDT component, then click within the schematic window at the desired placement location.



3. **Edit the LowpassDT component parameters.** Select the **LowpassDT** component by selecting it in the *SmartComponent* drop-down list box on the control window. Select the **Filter Assistant** tab. Change the response type by selecting the *Response Type* drop-down menu box. Choose **Chebyshev**.
4. **Design the LowpassDT component.** On the **Filter Assistant** tab, click the **Design** button. This will start the Filter Assistant and generate the design for the SmartComponent.
5. **Examine the LowpassDT filter design.** Select the component **LowpassDT** and click the **Push Into Hierarchy** button on the schematic toolbar. After examining the design, pop out of the SmartComponent by clicking the **Pop Out of Hierarchy** toolbar button.



6. **Analyze (simulate) the lowpass filter.** Select the **Simulation Assistant** tab page on the control window and click the **Simulate** button (accept default frequency/display specifications). This will start the Simulation Assistant and analyze the SmartComponent. The analysis results are shown here.

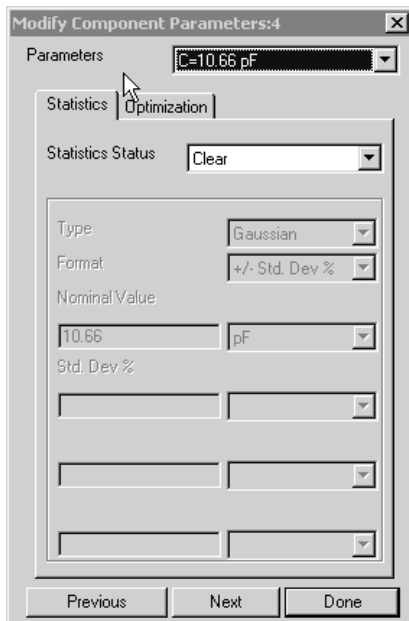


7. Close the lowpass filter analysis results window. Close the display window by choosing File > Close Window from the menu.

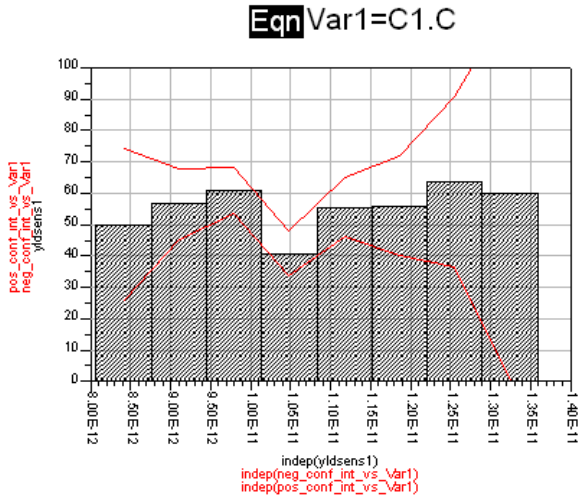
Analyze Yield of the Lowpass Filter

After the design process, the component sensitivities of the lowpass filter network can easily be checked using the **Yield Assistant**.

1. **Find components to analyze.** Select the **Yield Assistant** tab page on the control window. Choose the **View Components** button that lies beneath the **Statistical Components** list box, and then the **Modify Statistics/Optimization** button. The matching network will be displayed in the schematic along with the following dialog box.



2. **Choose components.** Select **Enabled** under **Statistics Status** and then press the **Next** button to move on to the next component. Repeat this until all three components in the network are enabled, and then choose **Done**.
3. **Analyze yield sensitivity of the network.** In the **Yield Assistant** tab, click **Automatically Set Frequencies > Simulate**. This will start the Yield Assistant and analyze the yield sensitivity to the selected component values. The sensitivity histogram for component C1 is shown here. You may select other pages in the display under the **Page** menu on the display window.



4. Close the filter yield analysis results window. Close the display window. Select **File > Close Window** from the menu.
5. Delete the LCLowpassDT SmartComponent. Select **Tools > Delete SmartComponent** from the DesignGuide control window menu.

Note This is different from the *Delete* button on the ADS schematic window toolbar.

Chapter 2: Using SmartComponents

This chapter describes how to use SmartComponents to design your filter.

Overview

This DesignGuide provides a large number of passive SmartComponents such as doubly terminated filters, singly terminated filters and bandpass transformers. SmartComponents are *smart* sub-network designs that can be placed into a schematic and provide the container for specification parameters and a schematic representation of the design. Several automated-assistants enable you to easily design, simulate (analyze) and check sensitivities of the 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 here.

There are two methods to display the desired SmartComponent palette.

- Open the Filter DesignGuide control window by selecting **DesignGuide > Filter > Filter Control Window**. Display the desired SmartComponent palette by selecting the **Component Palette** button from the control window toolbar or by selecting **View > Component Palette - All** from the control window menu.
- Select the desired SmartComponent palette from the Component Palette drop-down list box in the Schematic window toolbar (directly above the palette).

Placing SmartComponents

To place a SmartComponent:

1. Click on the desired component button in the SmartComponent palette.
2. Click within the Schematic window at the location you want the SmartComponent placed.
3. You may change the orientation of the SmartComponent before placement by selecting from the **Insert > Component > Component Orientation** commands or by repeatedly selecting **Rotate by -90** from the schematic toolbar.
4. The place component mode will remain active until you choose **End Command** from the schematic toolbar.

The available component palettes is as follows:

- *All* contains all of the SmartComponents.

Note When a SmartComponent is initially placed, a temporary component is used to initially place and specify the parameters for the SmartComponent. This component *does not* contain a subnetwork design. After the Design Assistant 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 Design Assistant will need to be used again to update the subnetwork design file.

Copying SmartComponents

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

Copying Within A 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

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.

Note All copied SmartComponents will initially refer to the same SmartComponent design. When the Design Assistant is used to perform a design operation, it will transform each copied SmartComponent into a unique SmartComponent design. A design operation is accomplished by launching the Design Assistant from the DesignGuide Control Window.

Editing SmartComponents

A SmartComponent's position, orientation, and parameters can be edited like any other component in ADS.

Position and Orientation

A SmartComponent is moved by dragging it to any location in the Schematic window. Its orientation is changed by following these steps.

1. Select **Edit > Advanced Rotate/Mirror > Rotate** from the Schematic window or select **Rotate Items** from the toolbar.
2. Click on the desired SmartComponent.
3. Rotate the component.
4. The rotate mode will remain active until you select the *End Command* from the toolbar.

Parameters

Parameters are changed by clicking on a SmartComponent parameter in the Schematic window and editing it or by double-clicking a component and editing the parameters in the component dialog box. Parameters may also be changed directly from the DesignGuide 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.

Delete From Current Design

A SmartComponent can be deleted from a design by selecting the component and pressing the **Delete** key, selecting **Delete** from the toolbar, or by selecting **Edit > Delete** from the schematic window. However, this does not remove the SmartComponent files from the project directory.

Delete From Current Project

To delete a SmartComponent and all associated files from your project, follow these steps.

1. From the DesignGuide control window or control toolbar, select the **Delete SmartComponent** button.
2. Click on the SmartComponent you want deleted. This will delete the SmartComponent from the current design and remove all of its files from your project.
3. The SmartComponent delete mode will remain active until you select the *End Command* from the schematic toolbar.

Delete Manually Using File System

You may 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_*, *SA_* and *SEN_*, contain the SmartComponent title, and end with *.ael* or *.dsn*.

Design, Analysis, and Yield

The DesignGuide contains several automated assistants that provide automatic design, analysis, and sensitivity simulation for the SmartComponents. The following assistants are available.

- **Filter (Design) Assistant.** The Filter Assistant is used to generate and update the design contained within a Filter SmartComponent. It invokes a synthesis engine that generates a design from the given specification. Refer to [“Filter Assistant” on page 3-1](#) for more information.
- **Simulation (Analysis) Assistant.** The Simulation Assistant is used to analyze the design contained within a SmartComponent. It creates a simulation circuit containing the SmartComponent, then performs a simulation. It can also

automatically display the results of the simulation. Refer to “[Simulation Assistant](#)” on page 3-4 for more information.

- **Yield Assistant.** The Yield Assistant is used to analyze the sensitivity of the network contained within a SmartComponent. It creates a yield analysis circuit containing the SmartComponent, then performs a simulation. Refer to the chapter “[Yield Assistant](#)” on page 3-6 for more information.
- **Display Assistant.** The Display Assistant is used to quickly display the performance of a SmartComponent. Display templates have been created for all of the SmartComponents. The display templates are preconfigured displays that provide a comprehensive look at the component’s performance. Refer to “[Display Assistant](#)” on page 3-11 for more information.

Stand-Alone SmartComponent Usage

Once SmartComponents are designed and tested, they can be used as stand-alone components. The Filter DesignGuide is not needed to use them in new designs unless you wish to modify or analyze them.

Using 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** from 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 will be listed in the Components list at the right of the Component Library window.
3. Select the desired SmartComponent in the Components list.
4. Place the desired SmartComponent into your design by clicking in the Schematic window at the location you wish it placed.
5. The insert mode will remain active until you select **End Command** from the toolbar.

Using an Existing SmartComponent in Any Project

A library of predesigned reusable SmartComponents can be easily created. This is done by placing the reusable SmartComponents in a project. This project can be

included in any project and its SmartComponents will be accessed using the Component Library. Follow these steps.

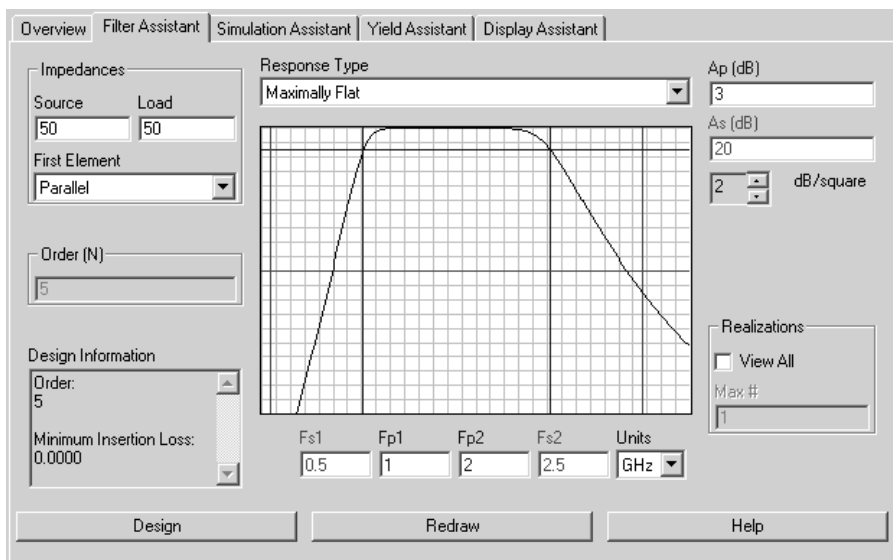
1. Select **File > Include/Remove Projects** from the main ADS window.
2. Select the project that contains the desired SmartComponent from the File Browser at the left of the *Include & Remove* window.
3. Choose the **Include** button to include the project in the hierarchy.
4. Chose the **OK** button.
5. Open the Component Library window by selecting **Insert > Component > Component Library** from the Schematic window or **Display Component Library List** from the toolbar.
6. Select the included project name under **All > Sub-networks** in the Libraries list at the left of the Component Library window.
7. Available components will be listed in the Components list at the right of the Component Library window.
8. Select the desired SmartComponent in the Components list.
9. Place the desired SmartComponent into your design by clicking in the Schematic window at the location you wish it placed.
10. The insert mode will remain active until you select **End Command** from the toolbar.

Chapter 3: Using Automated Assistants

This chapter describes the four Automated Assistants used to design, simulate, optimize, analyze, and transform SmartComponents.

Filter Assistant

The Filter Assistant is used to generate and update the design contained within a single SmartComponent from the given specifications. The Filter Assistant is accessed using the Filter DesignGuide control window. From the control window, full design control is enabled from the **Filter Assistant** tab. Component design operations can also be accomplished using the control window menu and toolbar. Any parameter change made from the **Filter Assistant** tab is reflected on the SmartComponent in the schematic.



To view a SmartComponent, select the desired SmartComponent from the *SmartComponent* drop-down list box in the upper right corner of the control window. The SmartComponent parameters are shown inside the **Filter Assistant** tab.

Impedances

- *Source* - source termination impedance(R_g), in Ohms. This value must be a real number greater than zero.
- *Load* - load termination impedance(R_l), in Ohms. This value must be a real number greater than zero.
- *First Component* - Either parallel or series. Refers to the first component in the network topology after designing. Parallel will design for minimal inductor values in the network. Series will design for minimal capacitor values in the network.

Order

The filter order is automatically calculated depending upon the parameters specified for all the response types except Bessel-Thomson and Gaussian. For these two responses a filter order must be input manually using the *Order (N)* text box.

Design Information

Design Information is not editable, but rather shows information about the filter.

- Order - is the filter order.
- Minimum Insertion Loss - shows the minimum insertion loss due to unequal terminations. The insertion loss is zero if terminations are matched.

Response Type

There are six filter response types that can be used for designing: Maximally Flat (Butterworth), Chebyshev, Elliptical, Inverse-Chebyshev, Bessel-Thomson, and Gaussian. A plot of the filter response is shown below the *Response Type* drop-down list box. Pressing the *Redraw* button will automatically scale and redraw the response plot.

Frequency

Frequency settings can be changed by either moving the vertical frequency slide bars on the response plot or by changing the values in the frequency text boxes. The

frequency text box captions dynamically change depending on the SmartComponent being used. For any lowpass or highpass network:

- Fp - passband corner frequency
- Fs - stopband frequency edge
- Units - used for both frequencies

For any bandpass or bandstop network:

- Fp1 - lower passband corner frequency
- Fp2 - upper passband corner frequency
- Fs1 - lower stopband frequency edge
- Fs2 - upper stopband frequency edge
- Units - used for all frequencies

Attenuation

Attenuation settings can be changed by either moving the horizontal attenuation slide bars on the response plot or by changing the values in the attenuation text boxes.

- Ap - passband attenuation, in dB
- As - stopband attenuation, in dB

Zooming in/out of the response plot is possible using the *dB/square* spin-box.

Realizations

These parameters tell the Filter DesignGuide how many networks to show after designing. If *View All* is not checked only one network will be synthesized. If *View All* is checked the number of shown networks can be specified in the *Max #* text box.

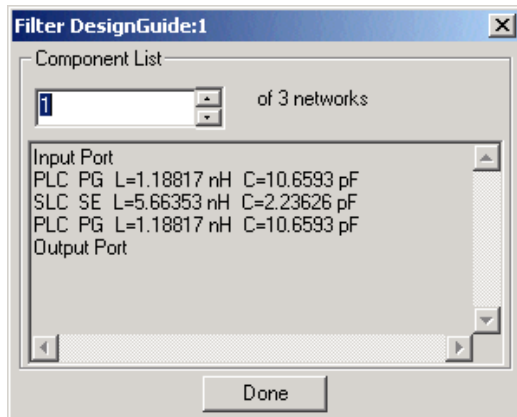
Design

The design is accomplished using one of the following methods.

- Push the **Design** button on the Design Assistant tab. The design progress is indicated on the tab page.

- Push the **Design** button on the Control Window toolbar.
- Select **Tools > Auto-Design** from the Control Window menu.

If multiple realizations for the filter network were found, a dialog box will appear.



All networks can be viewed using the *Component List* spin box. Each network will be viewed in two places:

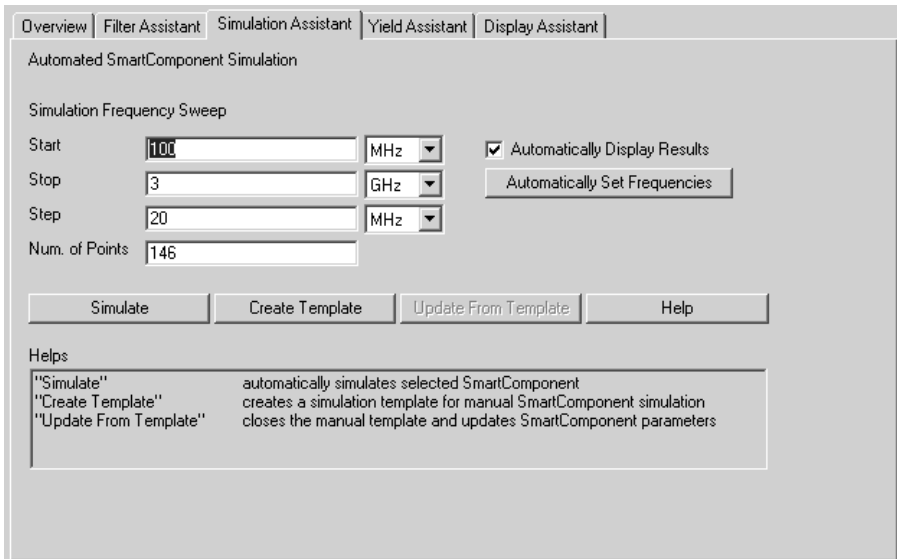
- Dialog Box - shows a text based description of the current network.
- Schematic Window - shows the actually drawing of the current network.

Choosing the *Done* button causes the dialog box to close. The last viewed filter network will become the subnetwork of the designed SmartComponent.

Simulation Assistant

The Simulation Assistant is used to analyze the design contained within a SmartComponent. It creates a simulation circuit around the SmartComponent, then automatically performs the appropriate simulation. If desired it will also automatically display the simulation results.

The Simulation Assistant is accessed using the *Filter DesignGuide* control window, where full simulation control is enabled from the *Simulation Assistant* tab. Basic simulation can also be accomplished using the control window menu and toolbar.



For all simulation operations, the selected SmartComponent is designed if necessary, a simulation schematic is created, the simulation is performed, and the results are displayed. The simulation frequency sweep must be specified on the *Simulation Assistant* tab in the control window as described in detail below.

Simulation Frequency Sweep

The simulation frequency sweep is specified on the Filter DesignGuide control window. While performing the simulation from the control window, select the *Simulation Assistant* tab and specify the sweep by entering the start frequency, stop frequency, and either frequency step size or number of points. The values entered are stored in the selected SmartComponent (as displayed in the *SmartComponent* drop-down list box) and will be recalled each time this SmartComponent is selected.

Automatically Display Results

If the *Automatically Display Results* box on the control window Simulation Assistant tab is selected, the simulation results will be automatically displayed upon completion of the analysis.

Starting the Simulation

The simulation may be accomplished using one of the following methods.

- Choose the **Simulate** button on the *Simulation Assistant* tab.
- Choose the **Simulate** button on the control window toolbar.
- Select **Tools > Auto-Simulate** from the control window menu.

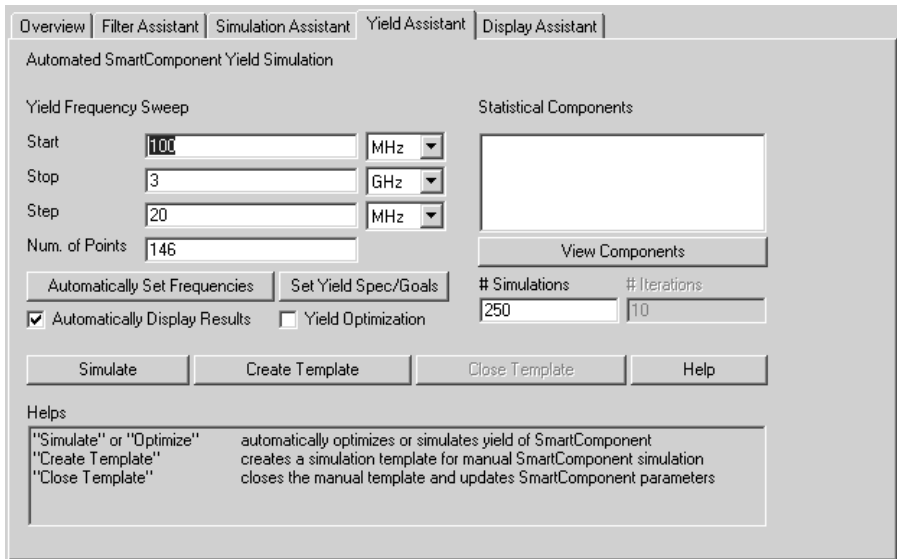
Simulation Templates

In some cases, it might be useful to manually simulate the SmartComponent. To generate a simulation schematic around the selected SmartComponent, press the *Create Template* button on the control window *Simulation Assistant* tab. You can examine or modify the simulation schematic, then manually start the simulation by selecting **Simulate > Simulate** from the Schematic window. When you are finished, pressing the **Update from Template** button on the *Simulation Assistant* tab will transfer any changes you have made to the SmartComponent on the simulation schematic to the original SmartComponent and redesign if necessary. You may also manually close the simulation schematic using **File > Close Design** from the Schematic window menu, although this will result in loss of any changes you have made to the SmartComponent.

Yield Assistant

The Yield Assistant is used to analyze the design sensitivities contained within a SmartComponent. It creates a yield analysis circuit containing the SmartComponent, then performs a simulation. By sweeping the component values for a selected set of components in the network, this analysis generates a probability density function of the performance given statistical variations of the component values. The probability that the performance will remain within the specified bounds is the yield of the network.

The Yield Assistant is accessed using the *Filter DesignGuide* control window, where full control is enabled from the *Yield Assistant* tab. Basic yield analysis can also be accomplished using the control window menu and toolbar.



The selected SmartComponent must be designed before yield analysis can be performed. The analysis proceeds by statistically sweeping the value of each selected component and analyzing the impact of this component value variation on the frequency response of the network.

Simulation Frequency Sweep

The simulation frequency sweep is specified on the *Yield Assistant* tab of the *Filter DesignGuide* control window. From this tab, specify the sweep by entering the start frequency, stop frequency, and either frequency step size or number of points. The values entered are stored in the selected SmartComponent (as displayed in the *SmartComponent* drop-down list box) and will be recalled each time this SmartComponent is selected.

Statistical Components

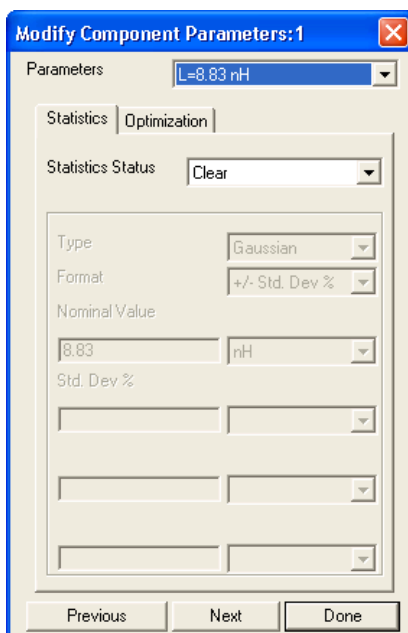
The *Statistical Components* list-box displays all components that will be statistically varied during simulation.

Pushing **View Components** will bring up a dialog box that shows the yield status of the components in the network.

Pressing **Modify Statistics/Optimization** brings up the dialog shown below to simplify the process of selecting components. The filter network will also be shown in the schematic with the currently selected component highlighted.

To sweep this component value statistically during the analysis, select **Enabled** in the *Statistics Status* box. You can then specify the parameters of the statistical sweep.

Once you have specified all parameters for a component, pressing **Next** proceeds to the next component in the network. The analysis allows for a maximum of 4 components to be selected at one time. Once you have finished specification, press **Done**.



Click **View Statistical Components** to view a summary of the statistical parameters for each selected component. Click **Modify Statistics/Optimization** to open the *Modify Component Parameters* dialog from this summary dialog to facilitate editing of the statistical sweep parameters.

The *# Simulations* parameter specifies the number of Monte Carlo simulations that will be used to estimate the statistical behavior of the network. Increasing the number of simulations will increase the statistical sample size and therefore provide a better estimate of the performance at the expense of increased computational time.

Yield Optimization

The network component values can also be optimized so that the performance is less sensitive to component value variations. This can be accomplished by selecting the *Yield Optimization* check box. The simulation will use the ADS engine to optimize the selected component values.

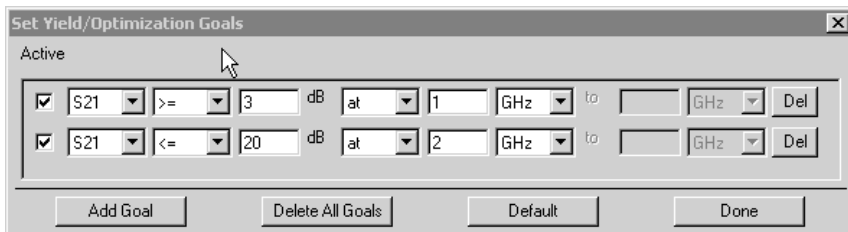
Specifying an appropriate set of *Yield/Optimization Goals* is a critical step in ensuring this optimization yields acceptable results. The *# Iterations* parameter available for Yield Optimization specifies the maximum number of optimization iterations that the simulation will perform to try to find the appropriate network component values.

Yield/Optimization Goals

Yield and Yield Optimization simulations require that a set of performance (yield) goals be specified for the network. The yield is defined as the probability that the network frequency response will satisfy these performance specifications given the statistical properties of the individual components.

Yield simulations use these goals to determine the percentage of networks that will satisfy the specifications given tolerances of the components. Yield Optimization simulations use the goals in determining suitable component values that maximize the yield.

Each component has a default set of goals depending on the type of response (lowpass, bandpass, etc.). You can modify these goals, including adding or deleting goals, by pressing the **Set Yield Spec/Goals** button on the *Yield Assistant* tab which will bring up the following dialog.



Each goal specifies the desired insertion loss performance of the network in dB and can represent a specification that the value stay above or below the stated level. The specification can be at a single point, or over a given frequency band. New goals can

be added using the **Add Goal** button. Individual goals can be deleted using the **Del** button to the right of the goal, or all of the goals can be deleted using the **Delete All Goals** button. The goals can be reset to their default values using the **Default** button. A goal will be used in the analysis only if the *Active* box at the left of the goal line is checked.

Automatically Display Results

If the *Automatically Display Results* box on the control window *Yield Assistant* tab is selected, the simulation results will be automatically displayed upon completion of the analysis.

Starting the Simulation

The yield analysis may accomplished using one of the following methods.

- Push the **Simulate** button on the *Yield Assistant* tab.
- Push the **Simulate Yield** button on the control window toolbar.
- Select **Tools > Auto-Simulate Yield** from the control window menu.

Yield Results

For each component (up to a maximum of 4) chosen for yield analysis, a yield sensitivity histogram will be displayed. The yield definition can be changed on the first page of the display by setting passband frequencies Fp_1 and Fp_2 as well as the maximum insertion loss at these frequencies, and stopband frequencies Fs_1 and Fs_2 as well as the minimum insertion loss at these frequencies. Other pages in the display show the overall statistics of the yield as well as the frequency response for each of the Monte Carlo simulations.

Yield Templates

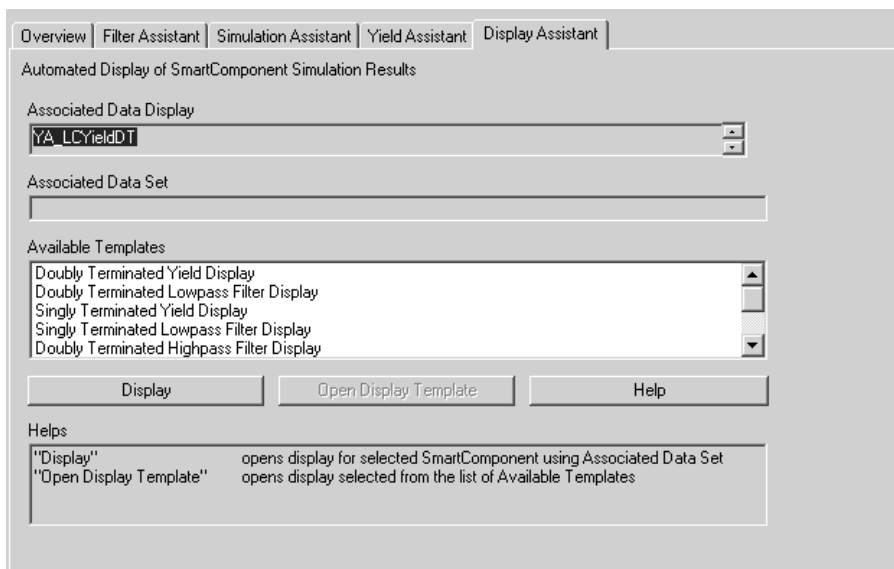
In some cases, it might be useful to manually simulate the SmartComponent. To generate a simulation schematic around the selected SmartComponent, press the **Create Template** button on the control window *Yield Assistant* tab. You can examine or modify the simulation schematic, then manually start the simulation by selecting **Simulate > Simulate** from the Schematic window. When you are finished, pressing the **Close Template** button on the *Yield Assistant* tab will return you to the original

design. You may also manually close the simulation schematic using **File > Close Design** from the Schematic window menu.

Display Assistant

The Display Assistant is used to easily and quickly display the performance of a SmartComponent. The display templates are preconfigured display files that provide a comprehensive look at the performance of the component. You can create your own displays or modify the included display templates using the built in features of Advanced Design System, but in most situations, the included display templates will provide all the information you need.

The Display Assistant is accessed using the Filter DesignGuide control window, where full display control is enabled from the *Display Assistant* tab. Basic display selection can also be accomplished using the control window menu and toolbar.

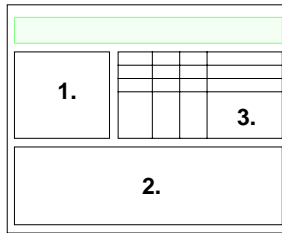


Display Template Features

The display templates opened by the Display Assistant have common features that are discussed here. For features unique to the display templates of some SmartComponents, refer to the chapter [Chapter 4, SmartComponent Reference](#).

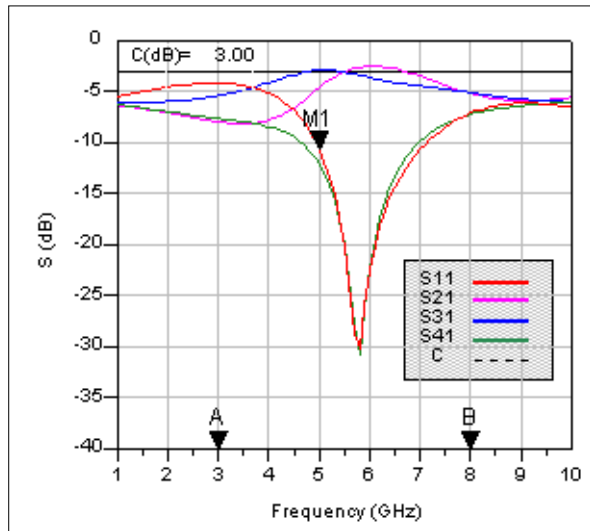
Basic Layout

Following is the basic layout of the display templates. Area one of the display template contains a graph of the most important parameters of the SmartComponent. Area two contains several graphs that give a comprehensive look at the component's performance. Area three contains a table listing the basic specifications and performance of the component.



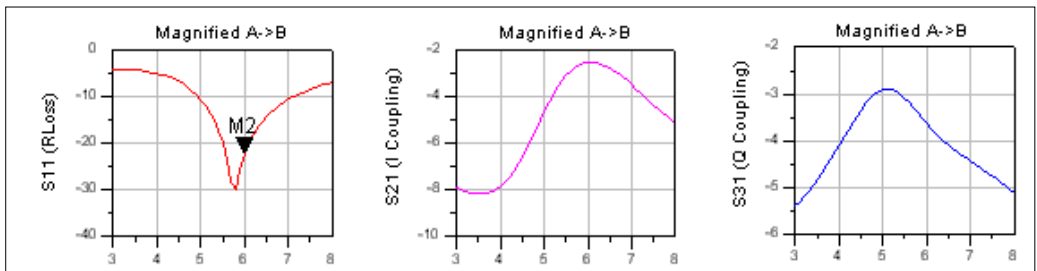
Typical Area One Graph

A typical graph from area one of a display template follows. The frequency range of the graph is determined by the Simulation Assistant. As you change the frequency range in the Simulation Assistant, this graph will update appropriately. The markers A and B are used to define the frequency range of the graphs in area two. This feature is used to zero in on the region of interest and obtain a comprehensive look at the component's performance. The marker M1 can be moved by dragging it with the mouse. The performance at the frequency given by M1 will be shown in the table in area three.



Typical Area Two Graphs

Typical graphs from area 2 of a display template are shown here. These graphs provide a quick, comprehensive look at the component's performance. Their frequency range is determined by the location of the "A" and "B" markers found in the main graph. Any markers such as M2 shown here can be moved by dragging them with the mouse. Performance criteria at the marker frequency will be displayed in the table in area three.



Typical Area Three Templates

A typical table from area three of a display template is shown here. The white rows show the desired specifications and important performance criteria for the component. The gray rows give the performance criteria at the user defined marker frequencies. The box below the table provides explanatory information for the table.

	F	S11	S21	S31	S41	PhaseD
Desired Center Frequency	5.00	-10.72	-4.64	-2.91	-12.15	84.94
Actual Center Frequency	5.80	-30.15	-2.64	-3.42	-30.81	90.23
Change/Worst A->B	5.00	-4.20	-8.16	-5.41	-7.22	70.77
Marker M1	5.00	-10.72	-4.64	-2.91	-12.15	84.94
Marker M2	6.00	-22.47	-2.54	-3.65	-22.25	89.63

F: Frequency (GHz) 1: Input Port 2: In-phase (I) Port 3: Quadrature (Q) Port 4: Isolated Port C: Desired Coupling PhaseD: I-Q Phase Difference	Note: Change/Worst A->B provides performance over the range from marker A to B. The change of F and PhaseD are given, and the worst case S-parameter values are given.
------------------------------------------------------------------------------------------------------------------------------------------------------------------	------------------------------------------------------------------------------------------------------------------------------------------------------------------------

Display Assistant Operation

Before using the Display Assistant, a valid dataset from a simulation of the selected SmartComponent must exist in the current project data directory. This simulation can be conveniently accomplished using the Simulation Assistant. Refer to the Simulation Assistant chapter for details on this step.

Opening a Display

To display results from a SmartComponent simulation using the control window, select the desired SmartComponent from the *SmartComponent* drop-down list box in the upper right corner of the control window. The display is then launched using one of the following methods.

- Choose the *Display* button on the **Display Assistant** tab.
- Choose the *Display* button on the control window toolbar.
- Select **Tools > Auto-Display** from the control window menu.

If no valid dataset exists for the selected SmartComponent, the *Display* button on the **Display Assistant** tab will be insensitive. If the toolbar or menu are used to try to display the results, a message will appear indicating that no dataset exists.

Display Templates

In some cases, it might be useful to use one of the display templates provided with the DesignGuide for other applications. To gain access to one of these templates, select the desired template from the *Available Templates* field and press the **Open Display Template** button on the control window *Display Assistant* tab. You can then insert a dataset of your choice using the dataset pull-down list box in the upper left corner of the display. You may find that some parameters in the display template are not defined in the selected dataset and may want to make appropriate modifications to the display. These changes can be saved using the commands in the display *File* menu.

Transformation Assistant

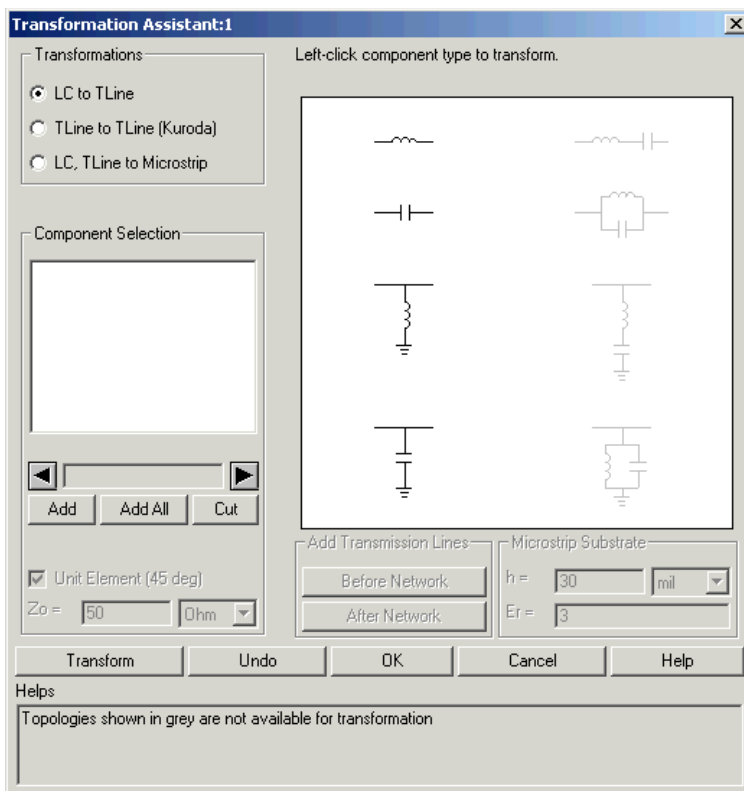
Once a Filter DesignGuide SmartComponent has been designed, the lumped inductors and capacitors can be transformed into equivalent distributed element counterparts using the Transformation Assistant. This feature enables you to quickly and easily transform an ideal filter topology to a form that is realizable for high-frequency systems.

Accessing the Transformation Assistant

The *Transformation Assistant* dialog box is accessed from the *Filter DesignGuide* control window, either by selecting **Tools > Distributed Element Transformations** from the control window menu or from the Toolbar.



When the Transformation Assistant is opened, the SmartComponent subnetwork appears in the schematic window and a dialog box is opened. The transformations are then accomplished using the controls on the dialog.



Selecting a Transformation Type

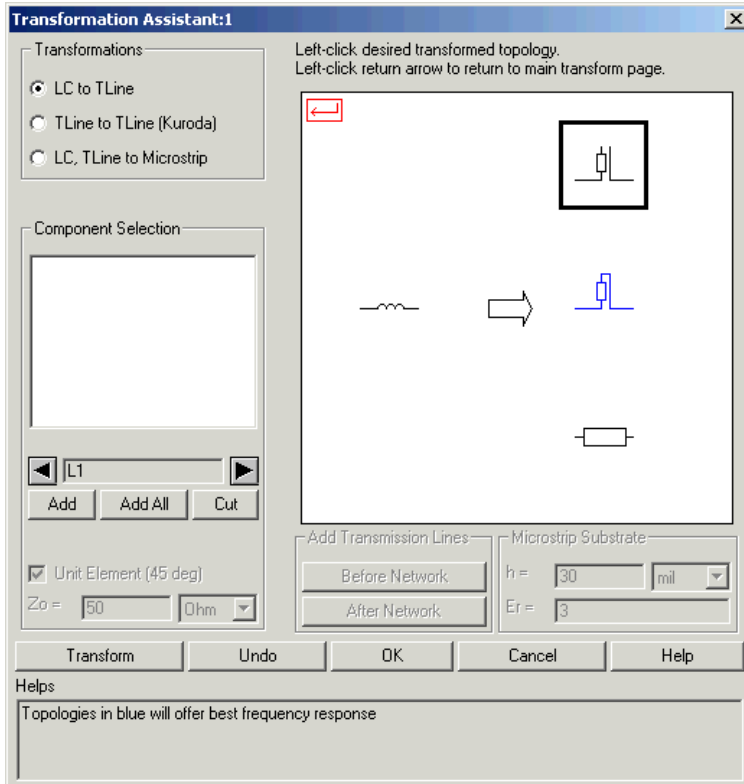
The type of transformation to be applied is selected from three options:

- *LC to TLine* - Transforms lumped inductors and capacitors to ideal transmission line elements. Eight different inductor/capacitor combinations can be transformed to different series lines, series stubs, or shunt stubs.
- *TLine to TLine (Kuroda)* - Apply Kuroda's identities in order to transform series short circuited stubs to shunt stubs that are realizable in microstrip and other printed transmission line technologies.

- *LC, TLine to Microstrip* - Transforms lumped inductors and capacitors as well as ideal transmission line structures to microstrip equivalent components. Application of this transformation requires a valid license for the Passive Circuit DesignGuide.

Once a transform has been selected, the graphical area displays the components that can be transformed using the current selection. Black components represent elements included in the original circuit available for transformation, while gray components represent elements not included in the original circuit.

From this graphical area, use the left mouse button to select one of the available component types. The graphical area will change to reveal the different distributed element equivalents available for substitution. The example below shows the transformations available when a series inductor circuit has been selected.



From this point, the type of equivalent network can be selected using the left mouse button from the available structures at the right of the graphics area. A box highlights the currently selected structure. Text at the bottom of the window changes as different selections are made, providing help for the particular transform selected.

Component Selection

Once the type of circuit component to be transformed is selected, the actual circuit elements to apply the transform to can be selected using the *Component Selection* tools. As the left and right arrows within this area are pushed, valid components within the original circuit will be highlighted, and their instance names (i.e. L1, C4) will appear in the text box on the *Transformation Assistant* dialog. The three buttons are used to select which specific components should be subject to the current transformation:

- **Add** - Add the currently selected component(s) to the transformation list.
- **Add All** - Add all circuit components of the appropriate type to the transformation list.
- **Cut** - Remove the currently selected (highlighted) item in the transformation list from the list.

Transformation Buttons

The buttons across the bottom of the dialog box are used to accomplish the transformation on the selected components.

- **Transform** - Apply the selected transform to the component in the transformation list.
- **Undo** - Undo the last performed transform. This button can be used repeatedly to undo all previous transformations.
- **OK** - Accept the current transformed circuit and close the dialog box. Once the transformed circuit has been accepted, transformations cannot be undone.
- **Cancel** - Close the dialog box and revert to the original, untransformed circuit.

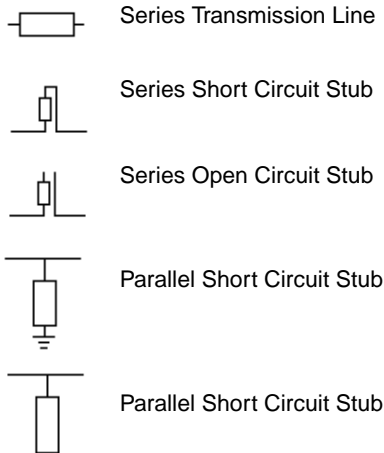
Changing Component Type

Once all transformations have been made on a specific component type (such as series inductor), performing a left mouse click on the red return arrow in the upper

left hand corner of the graphic area (or performing a right mouse click anywhere on the graphic area) can be used to return to the main component selection page. Another component type can then be selected, and the transformation steps can be repeated for this new selection.

Transmission Line Types

There are five basic transmission line elements that can be produced using the Transformation Assistant. These are identified as follows:



Additional Transformation Functions

Unit Element

For certain transformations, either the electrical length or characteristic impedance of the resulting transmission line must be specified by the user. If the **Unit Element** box is checked, the resulting transmission line will have an electrical length of 45 degrees and the characteristic impedance will be computed appropriately. If the **Unit Element** box is unchecked, then the **Characteristic Impedance (Z_0)** box will become active and the computation will use this characteristic impedance to compute the appropriate length.

Characteristic Impedance

The **Characteristic Impedance (Z_0)** box is used to specify the desired transmission line characteristic impedance for certain transformations. In cases where either the electrical length or the characteristic impedance can be specified, this box works in conjunction with the **Unit Element** box as discussed above. In certain other cases, this **Characteristic Impedance (Z_0)** box is used alone. For example, when adding lines to the front or end of a network as part of Kuroda's identities, the characteristic impedance of the transformation can be specified using this box.

Add Transmission Lines

As part of the *TLine to TLine* transformation, unit element (45 degree electrical length) transmission lines can be added to the front or end of the network. The characteristic impedance of these lines is specified using the **Characteristic Impedance (Z_0)** box. Such lines can be added as needed during the transformation process.

Note Addition of these lines will change the phase response and, if the characteristic impedance is not equal to the network terminal impedance, the magnitude response of the network.

Microstrip Substrate

When performing *LC*, *TLine to MLine* transformations, the microstrip substrate thickness (h) and relative permittivity (ϵ_r) must be specified. All microstrip elements within a design must share the same substrate parameters. The substrate parameters used in the final design will be the values that appear in the boxes after the final transformation step.

TLine to TLine Transforms (Kuroda Identities)

The TLine to TLine transforms are typically used to transform series short-circuited stubs to parallel open circuited stubs in preparation for implementation in planar transmission line technologies. These operations, however, only work on **Unit Element** lines with electrical lengths of 45 degrees. Therefore, when performing lumped to ideal distributed transformations, it is necessary to perform substitutions that conform to this **Unit Element** specification. Preferred stubs (highlighted in blue on the graphical area) as well as series transmission lines (transformed with the **Unit Element** box checked) will be able to be transformed in this way. When adding

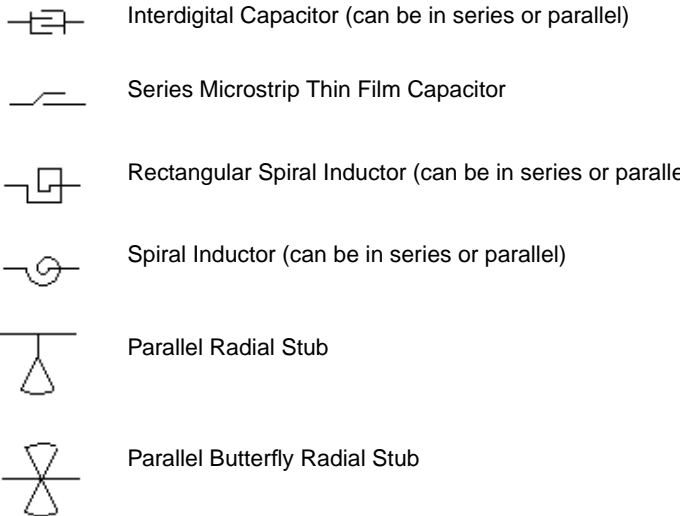
transmission line elements before or after the network, the electrical length will be 45 degrees and only the characteristic impedance must be specified.

Microstrip Transforms

Note This set of transforms is only available if a valid license for the Passive Circuit DesignGuide exists.

The *LC*, *TLine* to *MLine* transformations form a somewhat unique class of operations. This set of transformations takes lumped inductor/capacitor combinations as well as ideal series transmission lines and shunt transmission line stubs (obtained from the *LC* to *TLine* transformations), and converts them to microstrip. Note that series stubs cannot be used in this transformation since they cannot be realized in microstrip.

In addition to the standard transmission line topologies, certain lumped elements can be replaced with SmartComponents from the Passive Circuit DesignGuide. The available SmartComponents are as follows:



When making such substitutions, the design capabilities of the Passive Circuit DesignGuide are used to realize the topologies. In this case, however, the design procedure is approximate, and some tuning of the elements may be required before

the substituted device will offer the correct performance. In such cases, following completion of the transformation, push into the SmartComponent on the schematic window and open the Passive Circuit DesignGuide control window. The Simulation and Optimization Assistants in the Passive Circuit DesignGuide SmartComponent can then be used to quickly and efficiently tune the performance of each individual element.

Chapter 4: SmartComponent Reference

This chapter provides detailed information for all filter SmartComponents.

SmartComponent List

[LCLowpassDT \(Doubly Terminated Lowpass Filter\)](#)

[LCHighpassDT \(Doubly Terminated Highpass Filter\)](#)

[LCBandpassDT \(Doubly Terminated Bandpass Filter\)](#)

[LCBandstopDT \(Doubly Terminated Bandstop Filter\)](#)

[LCLowpassST \(Singly Terminated Lowpass Filter\)](#)

[LCHighpassST \(Singly Terminated Highpass Filter\)](#)

[LCBandpassST \(Singly Terminated Bandpass Filter\)](#)

[LCBandstopST \(Singly Terminated Bandstop Filter\)](#)

LCLowpassDT (Doubly Terminated Lowpass Filter)

Symbol



Parameters

F_p = passband corner frequency, in hertz

F_s = stopband frequency, in hertz

A_p = passband attenuation, in dB

A_s = stopband attenuation, in dB

N = order of filter, unitless

$ResponseType$ = type of filter response

$MinLorC$ = minimum inductance or capacitance

R_g = source impedance, in ohms

R_l = load impedance, in ohms

$MaxRealizations$ = maximum number of realizations during synthesis

Notes

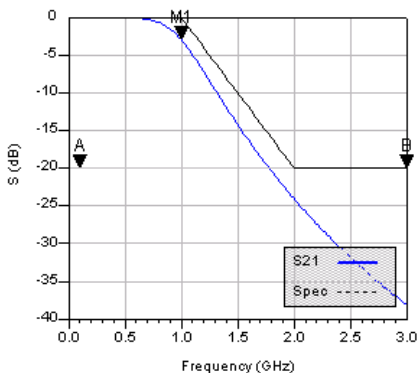
1. A doubly terminated lowpass filter provides a lowpass frequency response between the input (pin 1) and output (pin 2) ports. This component can have any real, non-zero impedance at either the source or load terminations.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter $ResponseType$. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

A maximally flat lowpass filter was designed for 3dB attenuation at 1 GHz and 20dB attenuation at 2 GHz with matched 50 ohm terminations. This resulted in a filter order of $N=4$.



DA_LCLowpassDT_Example
DA_LCLowpassDT1
Fp=1 GHz
Fs=2 GHz
Ap=3 dB
As=20 dB
N=4
Rg=50 Ohm
Rl=50 Ohm



LCHighpassDT (Doubly Terminated Highpass Filter)

Symbol



Parameters

F_s = stopband frequency, in hertz

F_p = passband corner frequency, in hertz

A_p = passband attenuation, in dB

A_s = stopband attenuation, in dB

N = order of filter, unitless

$ResponseType$ = type of filter response

$MinLorC$ = minimum inductance or capacitance

R_g = source impedance, in ohms

R_l = load impedance, in ohms

$MaxRealizations$ = maximum number of realizations during synthesis

Notes

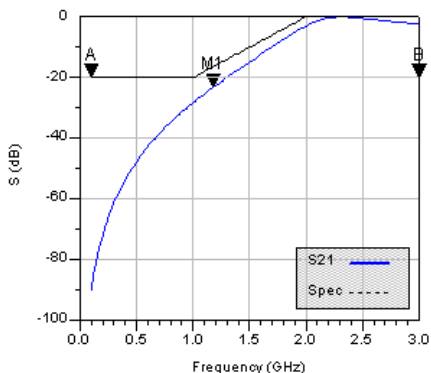
1. A doubly terminated highpass filter provides a highpass frequency response between the input (pin 1) and output (pin 2) ports. This component can have any real, non-zero impedance at either the source or load terminations.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter $ResponseType$. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

A chebyshev highpass filter was designed for 20dB attenuation at 1 GHz and 3 dB attenuation at 2 GHz with matched 50 ohm terminations. This resulted in a filter order of $N=3$.



DA_LCHighpassDT_Example
DA_LCHighpassDT1
Fs=1 GHz
Fp=2 GHz
Ap=3 dB
As=20 dB
N=3
Rg=50 Ohm
Rl=50 Ohm



LCBandpassDT (Doubly Terminated Bandpass Filter)

Symbol



Parameters

Fs1 = lower stopband frequency, in hertz

Fp1 = lower passband corner frequency, in hertz

Fp2 = upper passband corner frequency, in hertz

Fs2 = upper stopband frequency, in hertz

Ap = passband attenuation, in dB

As = stopband attenuation, in dB

N = order of filter, unitless

ResponseType = type of filter response

MinLorC = minimum inductance or capacitance

Rg = source impedance, in ohms

RI = load impedance, in ohms

MaxRealizations = maximum number of realizations during synthesis

Notes

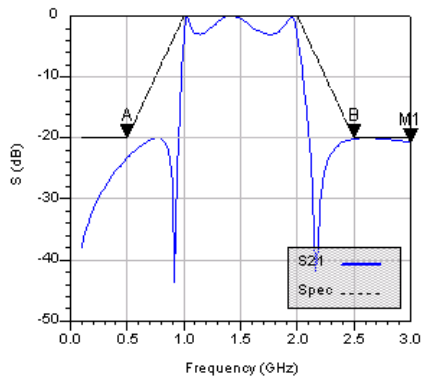
1. A doubly terminated bandpass filter provides a bandpass frequency response between the input (pin 1) and output (pin 2) ports. This component can have any real, non-zero impedance at either the source or load terminations.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter ResponseType. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

An elliptical bandpass filter was designed for 3dB attenuation between 1 GHz and 2 GHz and 20 dB attenuation below .5 GHz and above 2.5 GHz with matched 50 ohm loads. This resulted in a filter order of $N=4$.



DA_LCBandpassDT_Example
DA_LCBandpassDT1
Fs1=0.5 GHz
Fp1=1 GHz
Fp2=2 GHz
Fs2=2.5 GHz
Ap=3 dB
As=20 dB
N=3
Rg=50 Ohm
Rl=50 Ohm



LCBandstopDT (Doubly Terminated Bandstop Filter)

Symbol



Parameters

Fp1 = lower passband corner frequency, in hertz

Fs1 = lower stopband frequency, in hertz

Fs2 = upper stopband frequency, in hertz

Fp2 = upper passband corner frequency, in hertz

Ap = passband attenuation, in dB

As = stopband attenuation, in dB

N = order of filter, unitless

ResponseType = type of filter response

MinLorC = minimum inductance or capacitance

Rg = source impedance, in ohms

RI = load impedance, in ohms

MaxRealizations = maximum number of realizations during synthesis

Notes

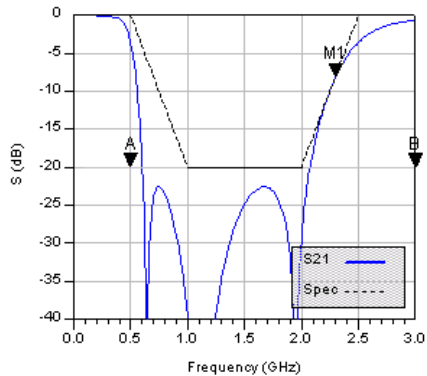
1. A doubly terminated bandstop filter provides a bandstop frequency response between the input (pin 1) and output (pin 2) ports. This component can have any real, non-zero impedance at either the source or load terminations.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter ResponseType. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

An Inverse-Chebyshev bandstop filter was designed for 20 dB attenuation between 1 GHz and 2 GHz and 3 dB attenuation below .5 GHz and above 2.5 GHz with matched 50 ohm loads. This resulted in a filter order of $N=4$.



DA_LCBandstopDT_Example
DA_LCBandstopDT1
Fp1=0.5 GHz
Fs1=1 GHz
Fs2=2 GHz
Fp2=2.5 GHz
Ap=3 dB
As=20 dB
N=4
Rg=50 Ohm
Rl=50 Ohm



LCLowpassST (Singly Terminated Lowpass Filter)

Symbol



Parameters

Fp = passband corner frequency, in hertz

Fs = stopband frequency, in hertz

Ap = passband attenuation, in dB

As = stopband attenuation, in dB

N = order of filter, unitless

ResponseType = type of filter response

MinLorC = minimum inductance or capacitance

Rg = source impedance, in ohms

Termination = open or short load termination

MaxRealizations = maximum number of realizations during synthesis

Notes

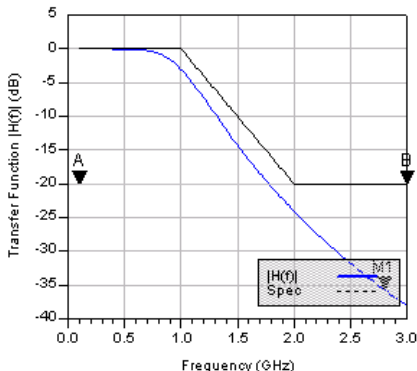
1. A singly terminated lowpass filter provides a lowpass frequency response between the input (pin 1) and output (pin 2) ports. This component will have a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter ResponseType. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

A maximally flat lowpass filter was designed for 3 dB attenuation at 1 GHz and 20 dB attenuation at 2 GHz with matched 50 ohm terminations. This resulted in a filter order of $N=4$.

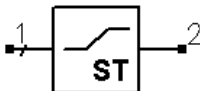


DA_LCLowpassST_Example
DA_LCLowpassST1
Fp=1 GHz
Fs=2 GHz
Ap=3 dB
As=20 dB
N=4
Rg=50 Ohm



LCHighpassST (Singly Terminated Highpass Filter)

Symbol



Parameters

F_s = stopband frequency, in hertz

F_p = passband corner frequency, in hertz

A_p = passband attenuation, in dB

A_s = stopband attenuation, in dB

N = order of filter, unitless

ResponseType = type of filter response

MinLorC = minimum inductance or capacitance

R_g = source impedance, in ohms

Termination = open or short load termination

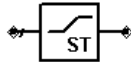
MaxRealizations = maximum number of realizations during synthesis

Notes

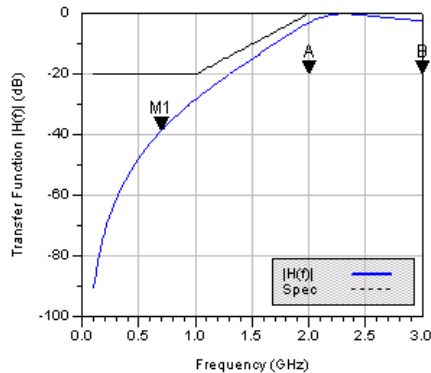
1. A singly terminated highpass filter provides a highpass frequency response between the input (pin 1) and output (pin 2) ports. This component will have a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter ResponseType. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

A chebyshev highpass filter was designed for 20 dB attenuation at 1 GHz and 3 dB attenuation at 2 GHz with matched 50 ohm terminations. This resulted in a filter order of $N=4$.

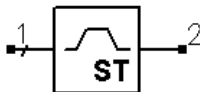


DA_LCHighpassST_Example
DA_LCHighpassST1
Fs=1 GHz
Fp=2 GHz
Ap=3 dB
As=20 dB
N=3
Rg=50 Ohm



LCBandpassST (Singly Terminated Bandpass Filter)

Symbol



Parameters

F_{s1} = lower stopband frequency, in hertz

F_{p1} = lower passband corner frequency, in hertz

F_{p2} = upper passband corner frequency, in hertz

F_{s2} = upper stopband frequency, in hertz

A_p = passband attenuation, in dB

A_s = stopband attenuation, in dB

N = order of filter, unitless

$ResponseType$ = type of filter response

$MinLorC$ = minimum inductance or capacitance

R_g = source impedance, in ohms

$Termination$ = open or short load termination

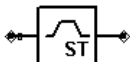
$MaxRealizations$ = maximum number of realizations during synthesis

Notes

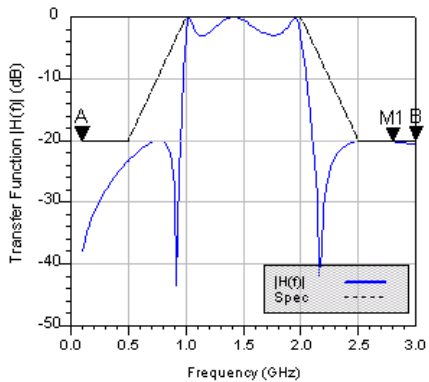
1. A singly terminated bandpass filter provides a bandpass frequency response between the input (pin 1) and output (pin 2) ports. This component will have a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter $ResponseType$. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

An elliptical bandpass filter was designed for 3 dB attenuation between 1 GHz and 2 GHz and 20 dB attenuation below .5 GHz and above 2.5 GHz with a 50 ohm source and shorted load. This resulted in a filter order of N=4.



DA_LCBandpassST_Example
DA_LCBandpassST1
Fs1=0.5 GHz
Fp1=1 GHz
Fp2=2 GHz
Fs2=2.5 GHz
Ap=3 dB
As=20 dB
N=3
Rg=50 Ohm



LCBandstopST (Singly Terminated Bandstop Filter)

Symbol



Parameters

Fp1 = lower passband corner frequency, in hertz

Fs1 = lower stopband frequency, in hertz

Fs2 = upper stopband frequency, in hertz

Fp2 = upper passband corner frequency, in hertz

Ap = passband attenuation, in dB

As = stopband attenuation, in dB

N = order of filter, unitless

ResponseType = type of filter response

MinLorC = minimum inductance or capacitance

Rg = source impedance, in ohms

Termination = open or short load termination

MaxRealizations = maximum number of realizations during synthesis

Notes

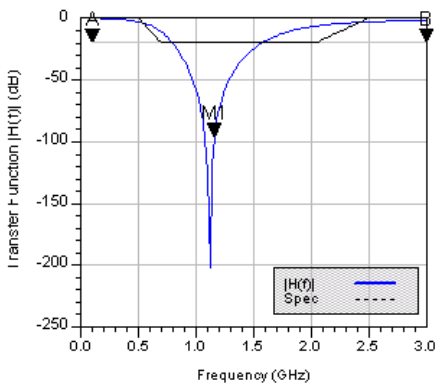
1. A singly terminated bandstop filter provides a bandstop frequency response between the input (pin 1) and output (pin 2) ports. This component will have a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (i.e. Chebyshev, Elliptical, etc.) depends on the parameter ResponseType. The transition bands of the frequency response depend on the filter order.
3. The filter order is either set manually or is calculated by the passband and stopband frequencies and attenuations.

Example

A maximally flat bandstop filter was designed for 20 dB attenuation between 1 GHz and 2 GHz and 3 dB attenuation below .5 GHz and above 2.5 GHz with a 50 ohm source and an open load. This resulted in a filter order of $N=7$.



DA_LCBandstopST_Example
DA_LCBandstopST1
Fp1=0.5 GHz
Fs1=1 GHz
Fs2=2 GHz
Fp2=2.5 GHz
Ap=3 dB
As=20 dB
N=4
Rg=50 Ohm



Index

A

automated assistants, overview, 1-5

C

control window, overview, 1-8

D

design flow, 1-3

design, creating a, 1-5

display assistant, overview, 3-11

F

filter assistant, overview, 3-1

L

LCBandpassDT, 4-6

LCBandpassST, 4-14

LCBandstopDT, 4-8

LCBandstopST, 4-16

LCHighpassDT, 4-4

LCHighpassST, 4-12

LCLowpassDT, 4-2

LCLowpassST, 4-10

P

palette access, 1-9

preferences, setting, 1-1

S

simulation assistant, overview, 3-4

SmartComponents

copying, 2-2

deleting, 2-3

editing, 2-3

overview, 2-1

placing, 2-1

standalone usage, 2-5

T

transformation assistant, overview, 3-15

Y

yield analysis, 1-13

yield assistant, overview, 3-6

