Agilent Technologies

Advanced Design System 2011.01

Feburary 2011
Filter DesignGuide

Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Mentor products and processes are registered trademarks of Mentor Graphics Corporation. * Calibre is a trademark of Mentor Graphics Corporation in the US and other countries. "Microsoft®, Windows®, MS Windows®, Windows NT®, Windows 2000® and Windows Internet Explorer® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Oracle and Java and registered trademarks of Oracle and/or its affiliates. Other names may be trademarks of their respective owners. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC. FLEXlm is a trademark of Globetrotter Software, Incorporated. Layout Boolean Engine by Klaas Holwerda, v1.7 http://www.xs4all.nl/~kholwerd/bool.html . FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg. QuestAgent search engine (c) 2000-2002, JObjects. Motif is a trademark of the Open Software Foundation. Netscape is a trademark of Netscape Communications Corporation. Netscape Portable Runtime (NSPR), Copyright (c) 1998-2003 The Mozilla Organization. A copy of the Mozilla Public License is at http://www.mozilla.org/MPL/ . FFTW, The Fastest Fourier Transform in the West, Copyright (c) 1997-1999 Massachusetts Institute of Technology. All rights reserved.

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

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

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

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

ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

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

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

UMFPACK 5.0.2 - UMFPACK Notice: The UMFPACK code was modified. Used by permission. UMFPACK Copyright: UMFPACK Copyright © 1995-2006 by Timothy A. Davis. All Rights Reserved. UMFPACK License: Your use or distribution of UMFPACK or any modified version of UMFPACK implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License

along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. UMFPACK Availability: http://www.cise.ufl.edu/research/sparse/umfpack UMFPACK (including versions 2.2.1 and earlier, in FORTRAN) is available at http://www.cise.ufl.edu/research/sparse . MA38 is available in the Harwell Subroutine Library. This version of UMFPACK includes a modified form of COLAMD Version 2.0, originally released on Jan. 31, 2000, also available at http://www.cise.ufl.edu/research/sparse . COLAMD V2.0 is also incorporated as a built-in function in MATLAB version 6.1, by The MathWorks, Inc. http://www.mathworks.com . COLAMD V1.0 appears as a column-preordering in SuperLU (SuperLU is available at http://www.netlib.org). UMFPACK v4.0 is a built-in routine in MATLAB 6.5. UMFPACK v4.3 is a built-in routine in MATLAB 7.1.

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

The HiSIM_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or STARC.

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

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

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

Restricted Rights Legend U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

Introducing the Filter DesignGuide

The Filter DesignGuide documentation provides an introduction to the Filter DesignGuide tool. 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 doubly terminated lowpass Chebyshev filter can be designed and verified, and a layout generated, in a few minutes.

The Filter DesignGuide provides *SmartComponents* and automated-assistants for the design and simulation. All SmartComponents can be modified. You simply select a SmartComponent and, with little effort, redesign or verify their performance. The "Using SmartComponents" section answers many common questions relating to DesignGuide use and the "SmartComponent Reference" describes each SmartComponent in detail. The section "Automated Design and Analysis" introduces Automated Assistants.

Step-by-Step Example

The step-by-step example takes you through the design, analysis and sensitivity simulation of a doubly terminated lowpass Chebyshev filter component. After completing this example, you should have a basic understanding of the DesignGuide and be ready to begin using the tool.

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* (usrguide) documentation.

Setting Up the Design Environment

Before you can use the Filter DesignGuide, you must set up the design environment by using these steps:

- Setting DesignGuide Preferences

- Opening a Workspace,

- Opening a Schematic Window,

- Opening the Filter DesignGuide,

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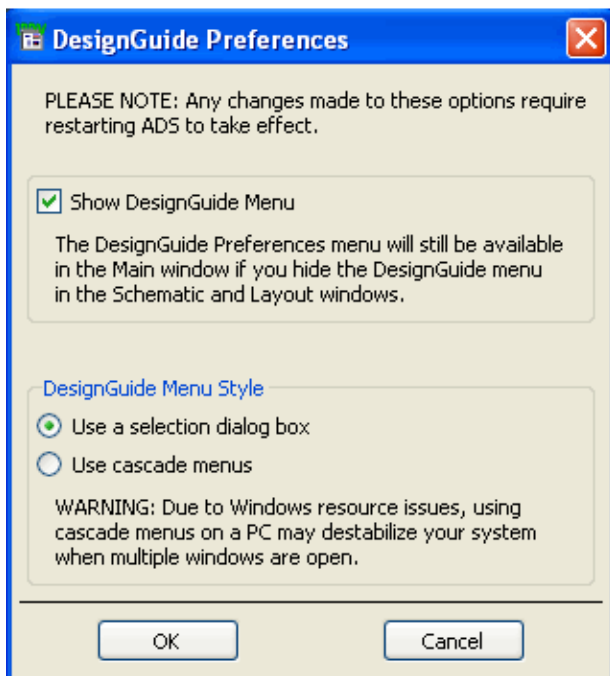
3. Close and restart the program for your preference changes to take effect.

> **ⓘ Note**
> On PC systems, Windows resource issues might limit the use of cascading menus. When multiple windows are open, your system could become destabilized. 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 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. 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 (cascading menus vs. dialog boxes).

Opening a Workspace

The ADS design environment is set up within a Workspace.

To create a new Workspace:

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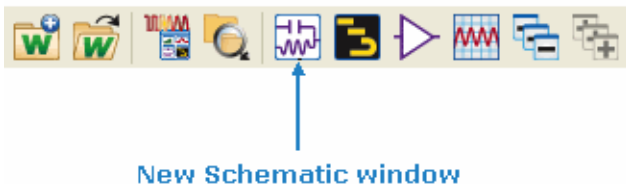

    Create a new workspace

2. In the dialog, define the location of the Workspace and assign a Workspace name. For details on creating or opening an existing Workspace, refer to *Using Workspace* (adstour).

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.

    New Schematic window

> ℹ️ **Note**
> Depending on how your ADS preferences are set, a Schematic window can appear automatically when you create or open a Workspace.

2. In the Schematic window, provide appropriate library and cell details, click **OK** to create a design named *Example*.

Opening the Filter DesignGuide

The **Filter** DesignGuide is accessed from the **Tools** menu or the **DesignGuide** menu in the Schematic window.

To open the Filter DesignGuide:

1. In the Schematic window, choose **DesignGuide** > **Filter** > **Filter Control Window** to open the Control window.

Using the Control Window

All DesignGuide 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 DesignGuide 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 the Filter Control window is selected from the **Tools** menu. From the **SmartComponent** drop-down list box, you can select any of the SmartComponents on the currently selected schematic window.
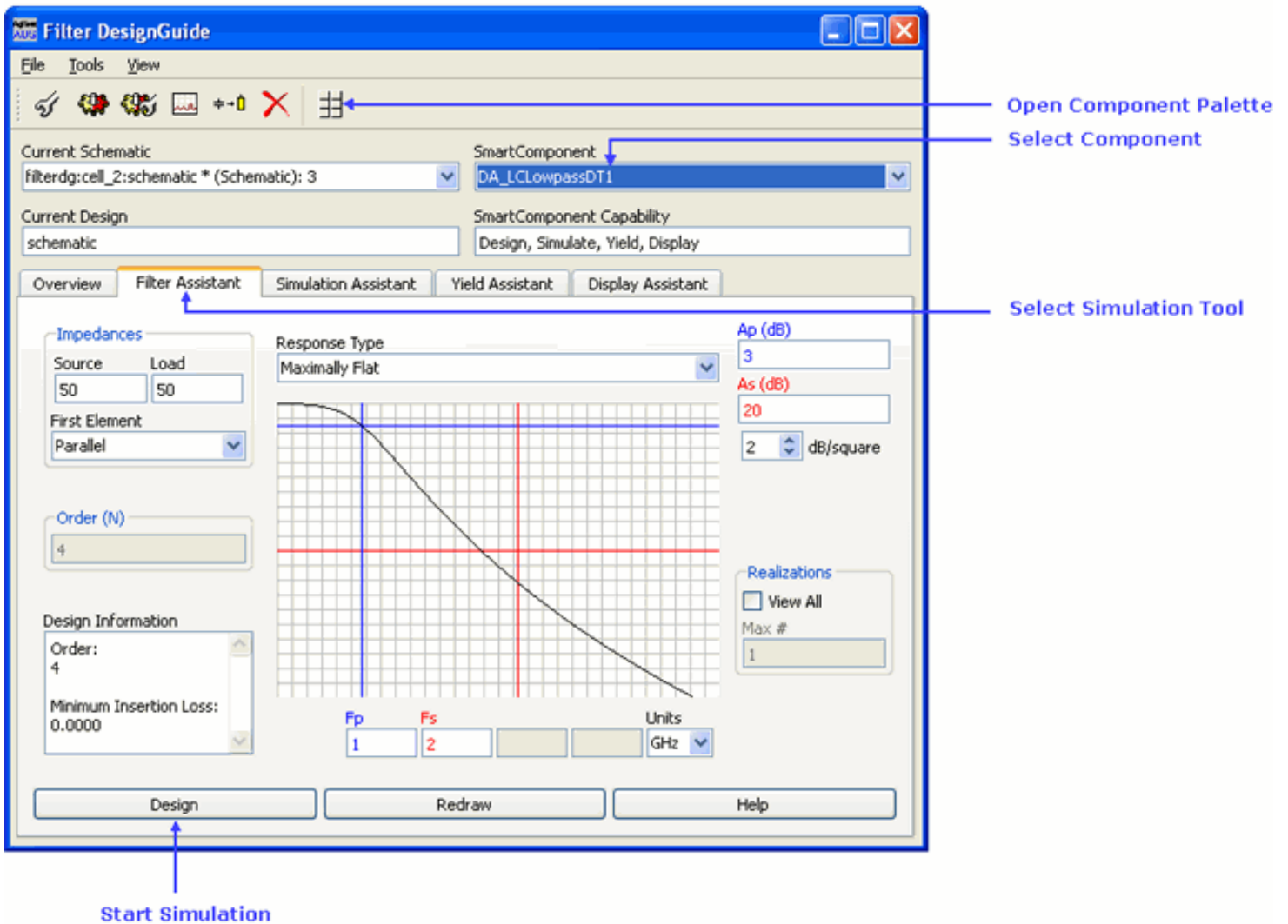
To close the Control window:

- 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 doubly terminated lowpass Chebyshev filter component. A component can be designed easily by using the default component parameter settings. Using the DesignGuide follows a normal design flow procedure:

- Select a component needed for your design from the component palette (Displaying the SmartComponent Palette) and placing the component in your design (Placing Example Component in the Design).

- Provide specifications (Changing SmartComponent Parameters).

- Design and analyze the component ([Designing the SmartComponent](#)) and [Analyzing Yield of the Network](#)).

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

Displaying the SmartComponent Palette

The program contains a SmartComponent palette, *Filter DG - All* , 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.
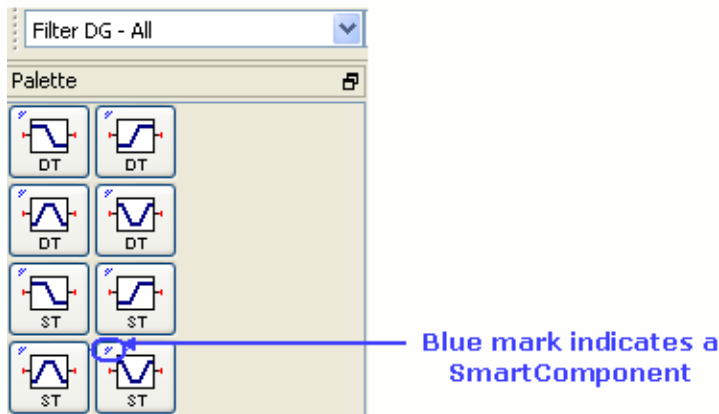
You can display the SmartComponent palettes in one of these ways:

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

Continue the example by selecting the *Filter DG - All* palette. The palette displays in the Schematic window.

Placing Example Component in the Design

To place the SmartComponent in the design:

1. Click **LCLowpassDT** on the component palette to select the component.
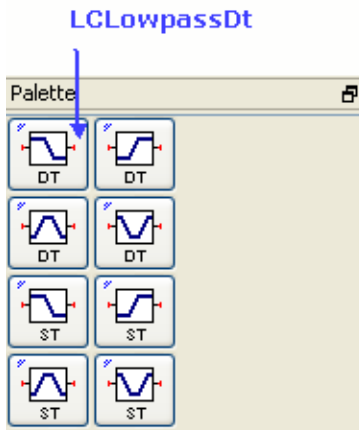
2. Click within the schematic window to place the component.

- 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 **Edit** menu in 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 DesignGuide 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 DesignGuide must be used again to update the subnetwork design file.

Changing SmartComponent Parameters

Parameters can be changed directly from the Control window.

To edit the LCLowpassDT component parameters:

1. In the Control window, select the **LCLowpassDT** ** component from the SmartComponent drop-down list. This ensures all changes are referenced to this component.
2. Select the **Filter Assistant** tab. Change the response type by selecting the Response Type drop-down menu box. Choose **Chebyshev**.

> **ⓘ 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.
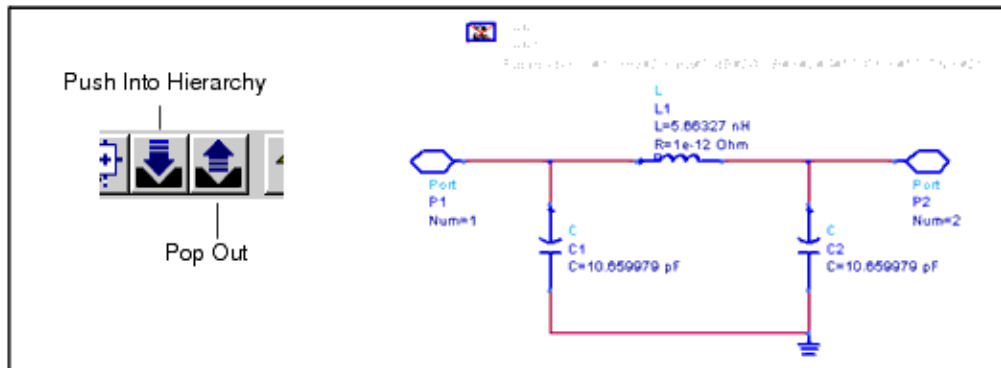
To start the simulation:

1. On the Filter Assistant tab, click **Design** to start a simulation and generate the design for the SmartComponent.

Examining the SmartComponent Design

You can look at the details of the autogenerated design inside the SmartComponent's subnetwork.
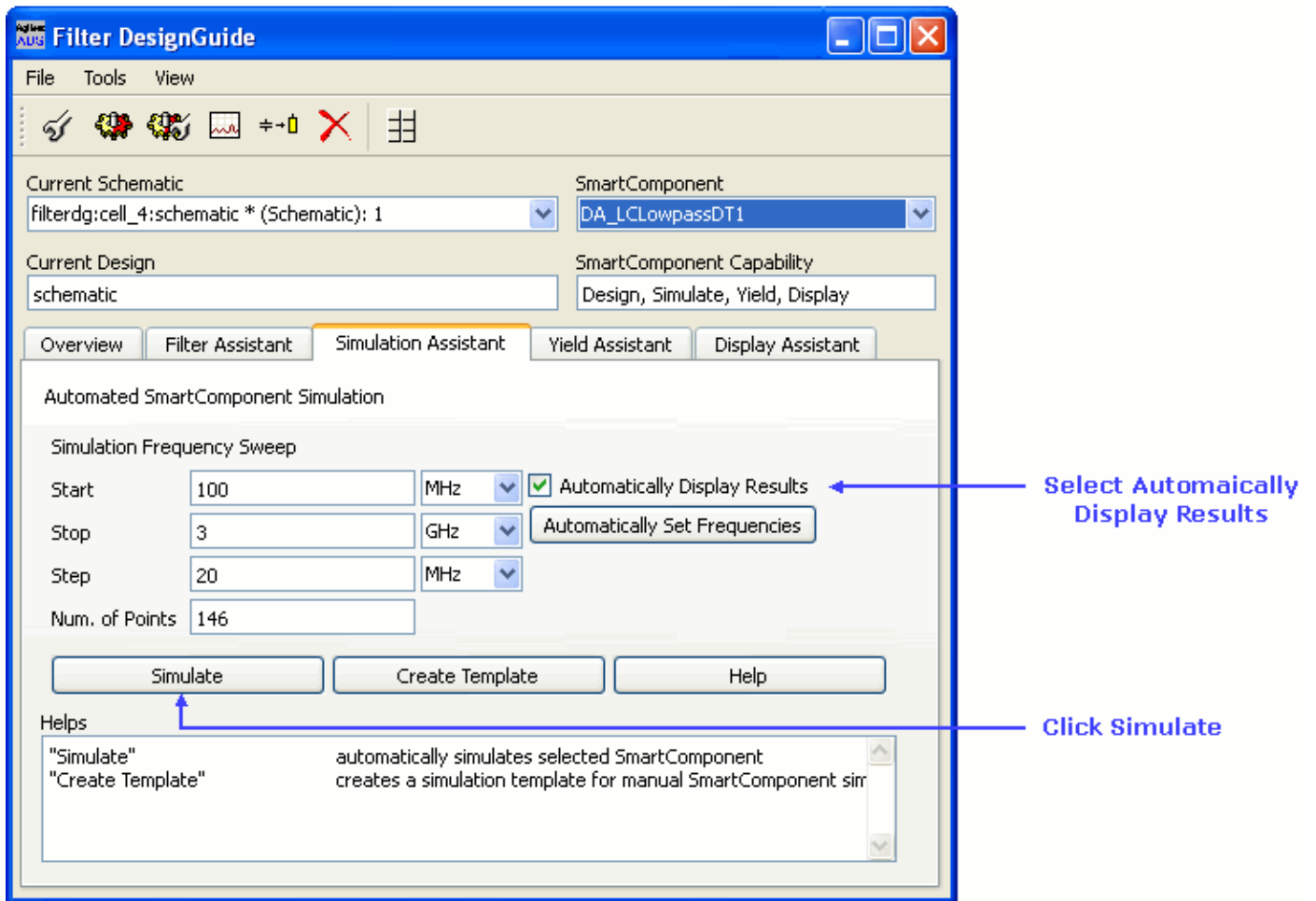
To examine the component's subnetwork:

1. Select the component **LCLowpassDG**.
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.

To analyze (simulate) the network:

1. Select the **Simulation Assistant** tab on the Control window.

2. Confirm that **Automatically Display Results** is active.
3. Accept default frequency/display specifications. Click **Simulate** to start the analysis. Part of the analysis results are shown below.
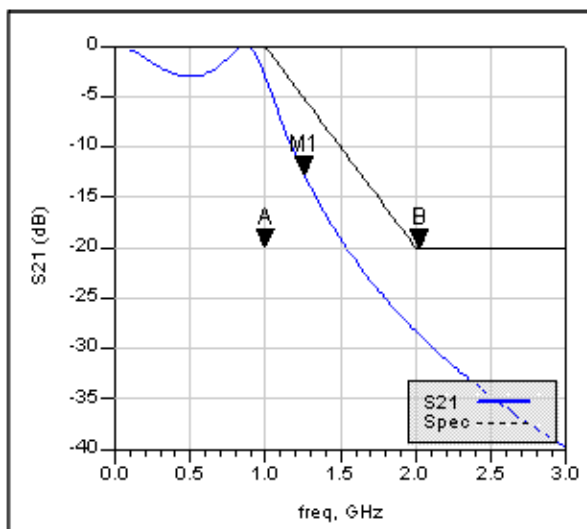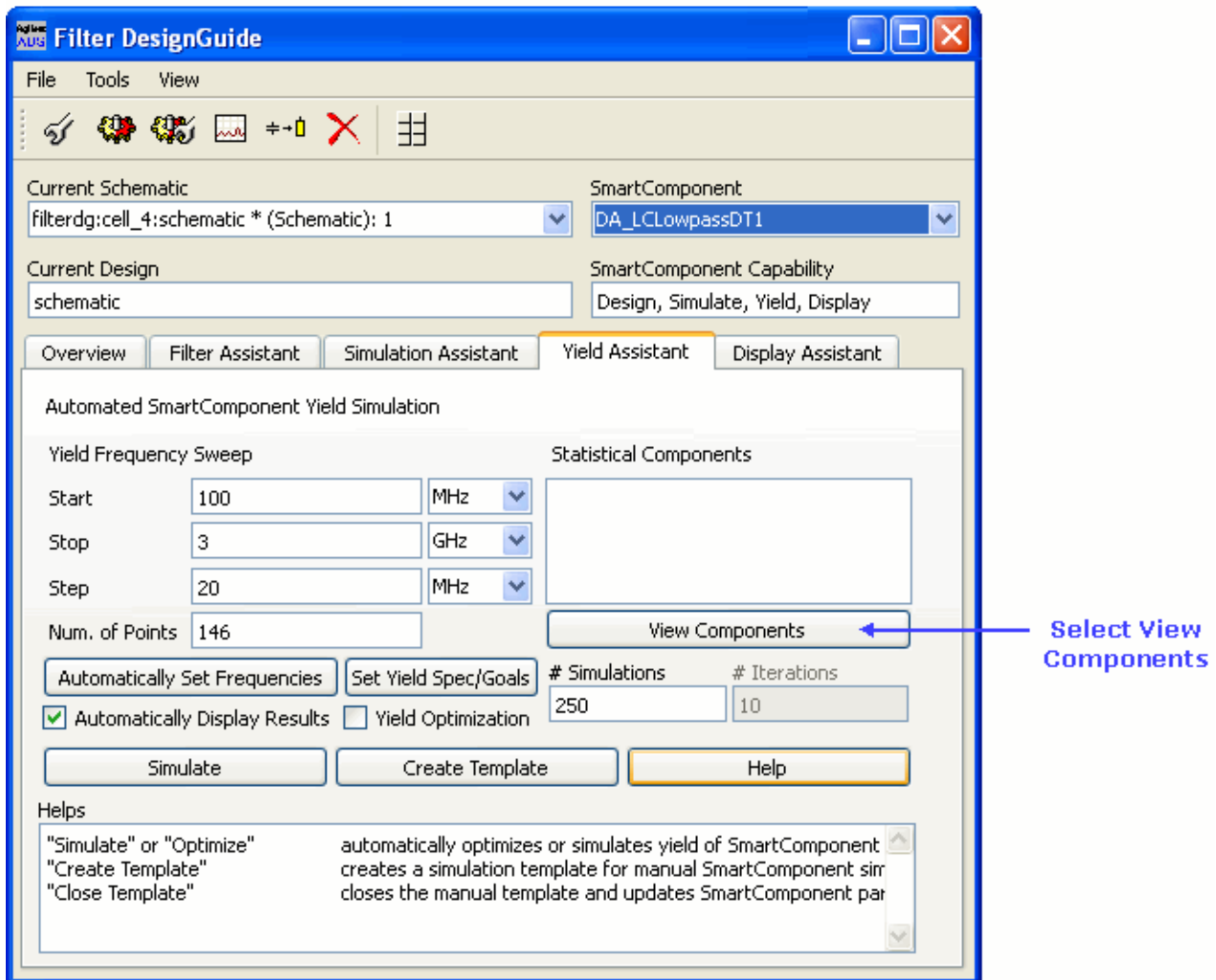

Closing the Results Window

- To close the display window, choose **File** > **Close Window** from the menu.
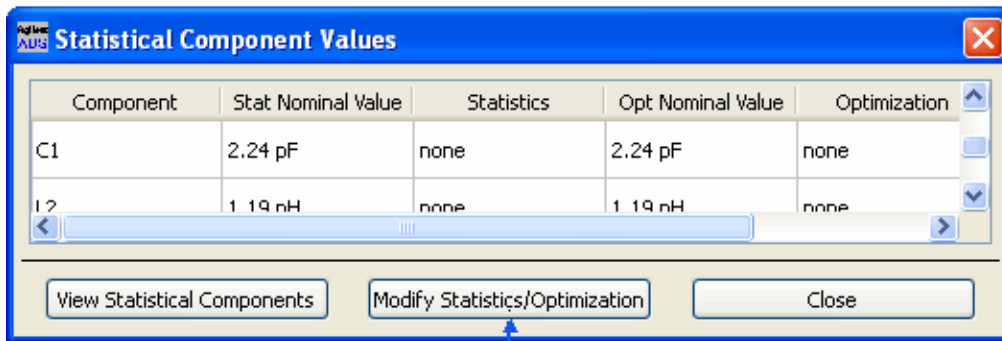
Analyzing Yield of the Network

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

To find components to analyze:

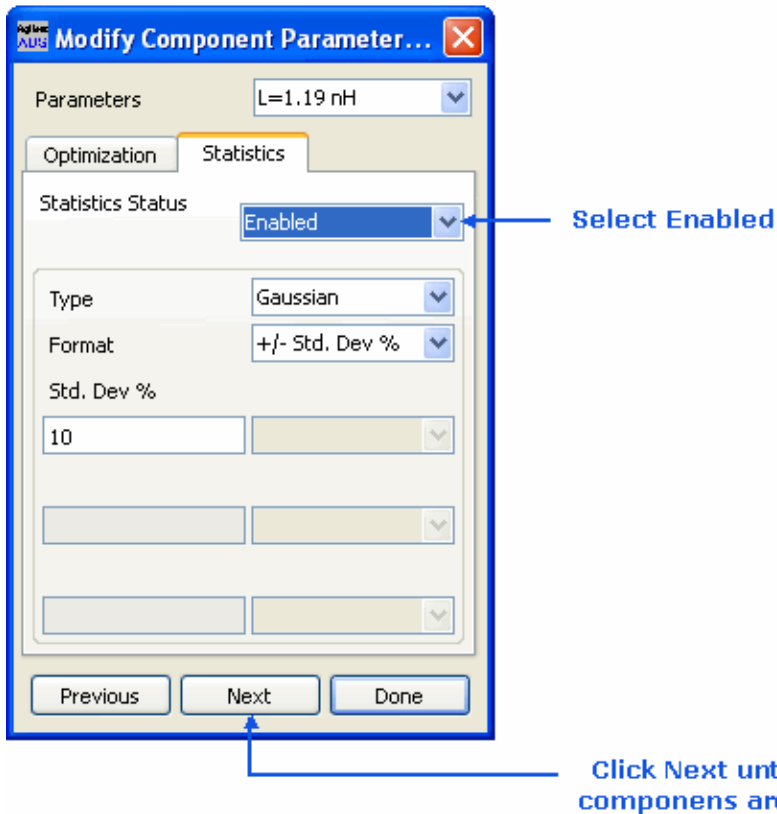1. Select the **Yield Assistant** tab on the Control window.

2. Click **View Components**. Then click **Modify Statistics/Optimization**. The network displays in the schematic along with a dialog box.

To choose components:

1. Choose **Enabled** from the **Statistics Status** drop-down menu.

2. Click **Next** to move on to the next component. Repeat this until all 3 components in the network are enabled, and then click **Done**.
3. Click **Close** in the Statistical Component Values dialog box.

To analyze yield sensitivity of the network:

1. In the **Yield Assistant** tab, click **Automatically Set Frequencies**.
2. Select by **Simulate** to start the Yield Assistant and analyze the yield sensitivity to the selected component values. The results display automatically. The sensitivity histogram for component C1 is shown here.
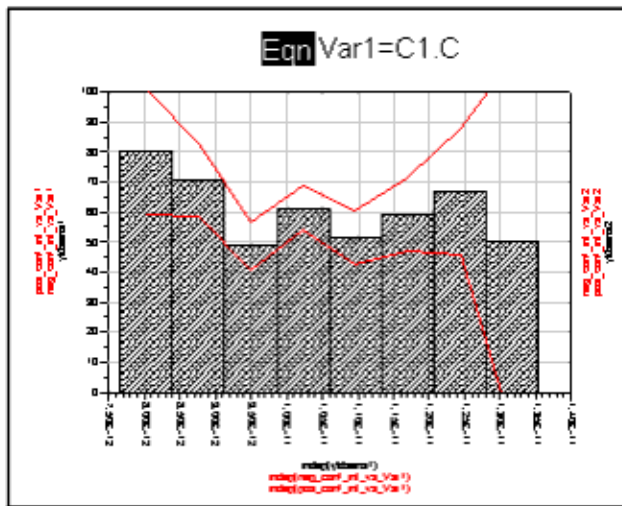
3. Choose **File** > **Close Window** from the menu to close the display window.

Deleting the SmartComponent

- To delete the LCLowpassDT SmartComponent, choose **Tools** > **Delete SmartComponent** from the Control window.

> ℹ️ **Note**
> The *Delete* button on the Control window is different from the *Delete* button on the ADS schematic window toolbar.

This completes the step-by-step example.

Using SmartComponents in Filter DesignGuide

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 provide the container for specification parameters and a schematic representation of the design when place into a design. The tool provides automated design and analysis for these SmartComponents.

Placing and Editing 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.

> 🛈 **Note**
> For help with ADS basic features, refer to the *Schematic Capture and Layout* (usrguide) documentation.

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.

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

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

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

You can modify the specifications in one of these ways:

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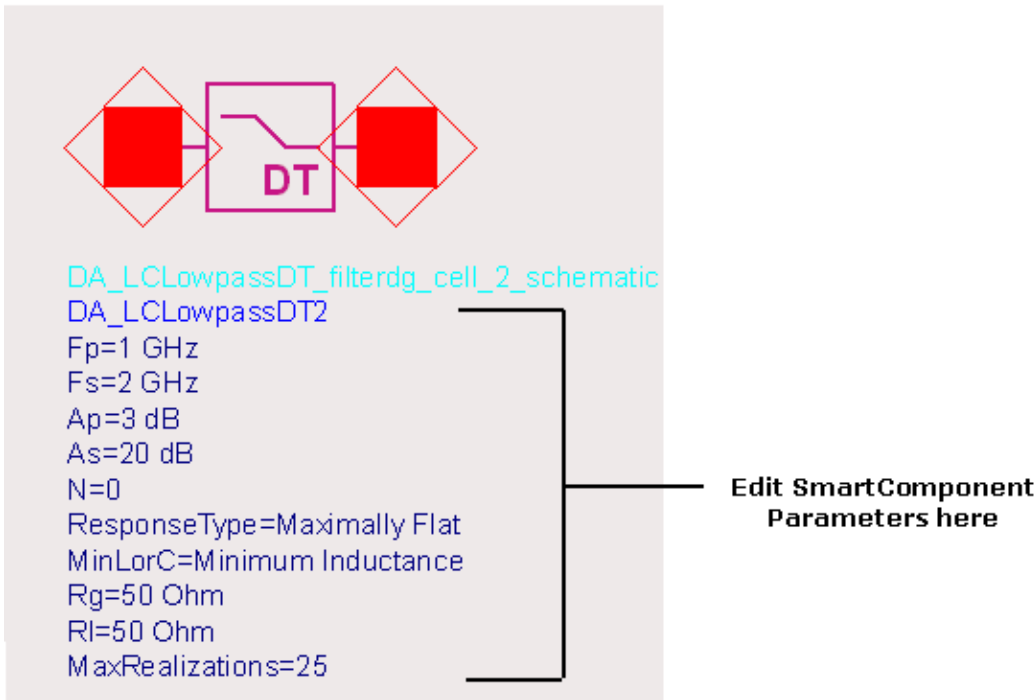
The LCLowpassDT Component

The SmartComponent design (schematic) can be viewed by pushing into the SmartComponent's subnetwork. See *Examining the SmartComponent Design* (dgfilter).

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 Filter 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 Workspace directory. To delete files from the Workspace, see Deleting from Current Workspace.

Deleting from Current Workspace

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

1. In the Schematic window, select the SmartComponent.
2. In the Control window, select **Tools** > **Delete SmartComponent.** or on the toolbar, click **Delete** . This deletes the SmartComponent from the current design and removes all of its files from your Workspace. 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 respective library of a Workspace.

Using SmartComponents as Standalone Components

After SmartComponents are designed and tested, they can be used as standalone components. The Filter DesignGuide is not needed to use them in new designs unless you want to modify or analyze them.

Using an Existing SmartComponent Within the Same Workspace

To use an existing SmartComponent within the same Workspace:

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 Library name under **All 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. .

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 to place. The insert mode remains active until you click **End Command**.

Using an Existing SmartComponent in Any Workspace

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

To use an existing SmartComponent in any Workspace:

1. Open the Workspace where the SmartComponent needs to be inserted.
2. Select the Workspace that contains the SmartComponent from list in the Include & Remove window.
3. Click **Include** to include the Workspace 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 Workspace 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 where you wish to place the component. The insert mode remains active until you click **End Command**.

Using Automated Assistants in Filter DesignGuide

This section describes the Automated Assistants available in this DesignGuide.

- Filter Assistant is used to generate and update the design contained within a SmartComponent's schematic design for filter networks.

- Simulation Assistant is used to automatically perform a simulation of a SmartComponent.

- Yield Assistant is used to automatically perform a simulation of a SmartComponent's network sensitivity.

- Display Assistant is used to automatically display the analysis results generated using the Simulation Assistant.

- Transformation Assistant is provides an interactive environment for transforming lumped elements to transmission lines within a designed SmartComponent.
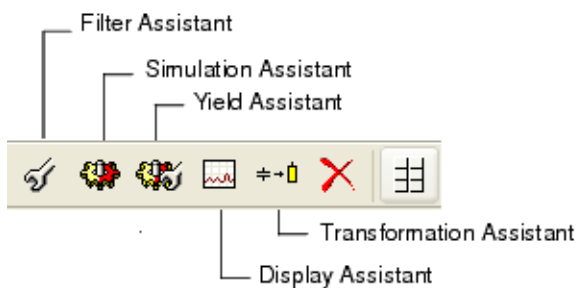
Filter Assistant

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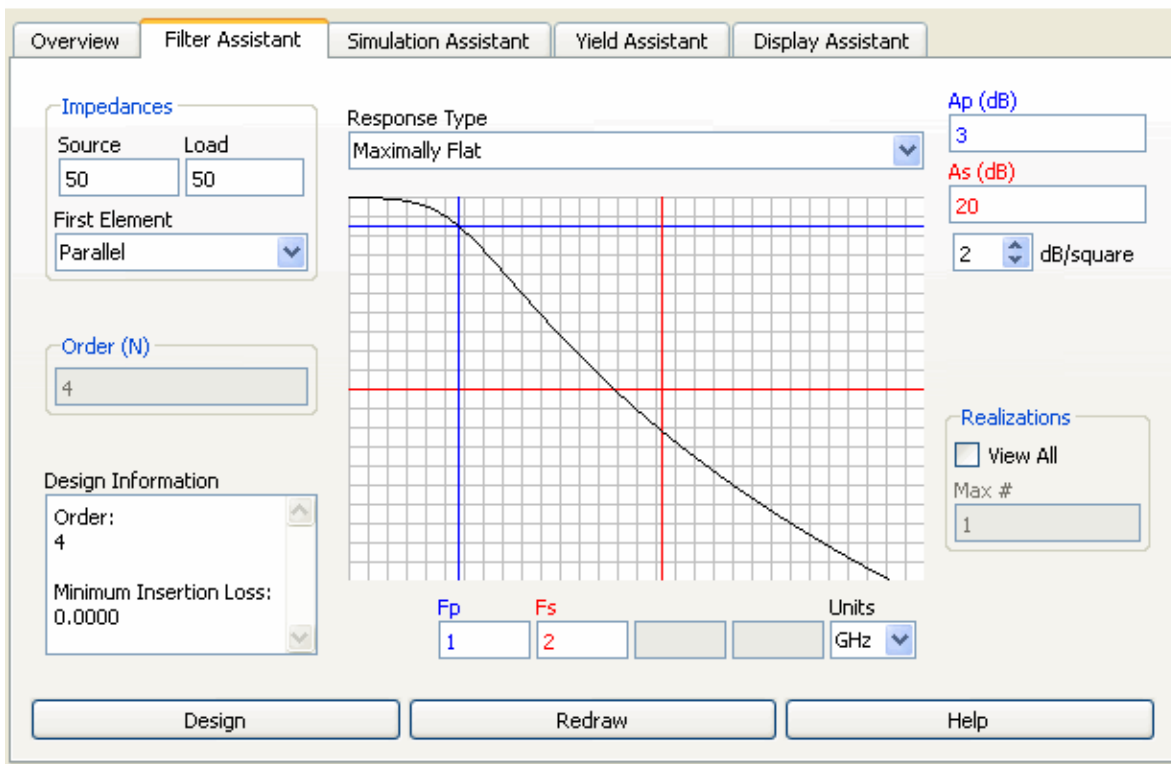
To view a SmartComponent, select the SmartComponent from the **SmartComponent** drop-down list box in the upper right corner of the Control window. The SmartComponent parameters are shown inside the **Filter Assistant** tab.

Impedances

- Source - Source termination impedance(Rg), in Ohms. This value must be a real number greater than zero.
- Load - Load termination impedance(Rl), 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 designs for minimal inductor values in the network. Series designs 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

Six filter response types can be used for designing: Maximally Flat (Butterworth), Chebyshev, Elliptical, Inverse-Chebyshev, Bessel-Thomson, and Gaussian. A plot of the filter response is show below the **Response Type** drop-down list box. Pressing the **Redraw** button automatically scales and redraws 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 of 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 is 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 these methods.

- Click **Design** on the Filter Assistant tab. The design progress is indicated on the tab page.
- Click **Design** on the Control window toolbar.
- Choose **Tools > Auto-Design** from the Control window menu.

If multiple realizations for the filter network were found, a dialog box appears.
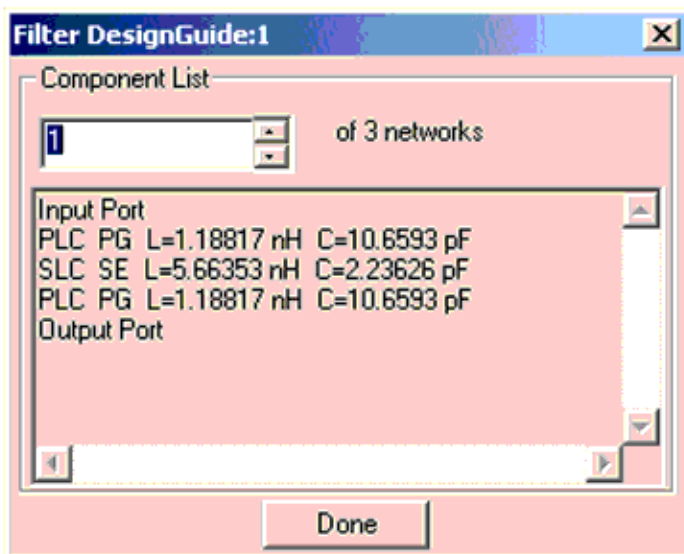
All networks can be viewed using the Component List spin box. Each network can be viewed in two places:

- *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 becomes the subnetwork of the designed SmartComponent.

Simulation Assistant

The Simulation Assistant is used to analyze the design contained within a SmartComponent. The Simulation Assistant creates a simulation circuit around the SmartComponent, then automatically performs the appropriate simulation. If set, the tool displays the simulation results automatically.

The Simulation Assistant is accessed using the *Filter DesignGuide* Control window, where full simulation control is enabled from the *Simulation Assistant* tab. Also, basic simulation can be accomplished using the Control window menu and toolbar.
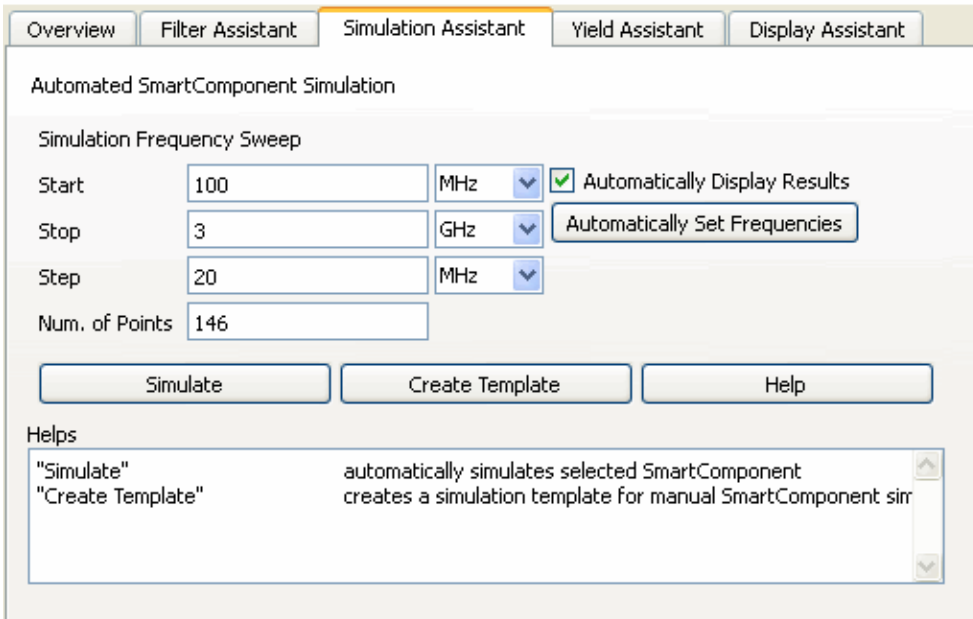
For all simulation operations, the selected SmartComponent is designed if necessary, a simulation schematic is created, the simulation is performed, and the results are displayed. The simulation frequency sweep must be specified on the *Simulation Assistant* tab in the Control window as described in detail in section Simulation Frequency Sweep.

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 is recalled each time this SmartComponent is selected.

Automatically Display Results

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

Starting the Simulation

The simulation can accomplished using one of the these methods.

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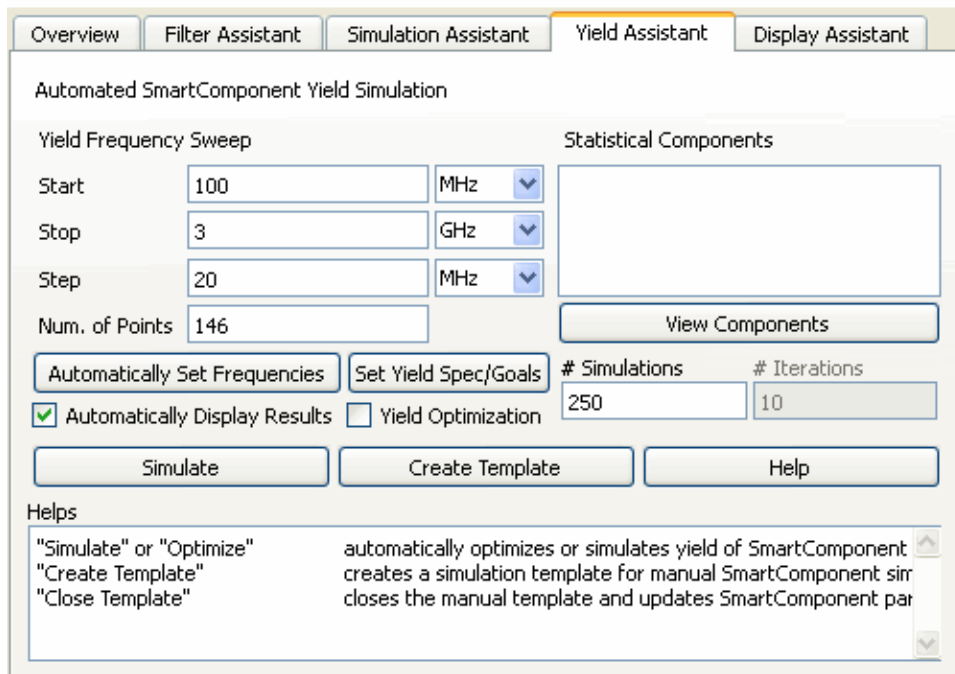- Choose *Tools > Auto-Simulate* from the Control window menu.

Simulation Templates

In some cases, you can simulate the SmartComponent manually. 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.

Yield Assistant

The Yield Assistant is used to analyze the design sensitivities contained within a SmartComponent. The Yield Assistant 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 remains 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. Also, basic yield analysis can 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 are recalled each time this SmartComponent is selected.

Statistical Components

The Statistical Components list-box displays all components that are statistically varied during simulation.
Clicking **View Components** opens a dialog box that shows the yield status of the components in the network.
Clicking **Modify Statistics/Optimization** opens the Modify Component Parameters dialog to simplify the process of selecting components. The filter network is displayed in the schematic with the currently-selected component highlighted.
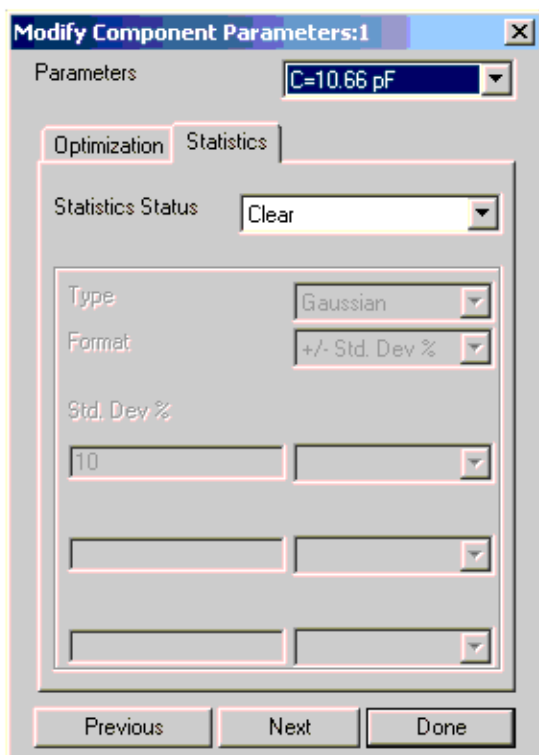
To sweep this component value statistically during the analysis, select **Enabled** in the Statistics Status box. Then specify the parameters of the statistical sweep.

After you have specified all parameters for a component, clicking **Next** proceeds to the next component in the network. You can select a maximum of 4 components at one time. After you have finished specification, click **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 are used to estimate the statistical behavior of the network. Increasing the number of simulations increases the statistical sample size and therefore provides 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 uses 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 performs 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 satisfies these performance specifications given the statistical properties of the individual components.
Yield simulations use these goals to determine the percentage of networks that satisfies 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 clicking *Set Yield Spec/Goals* on the Yield Assistant tab to display the dialog.
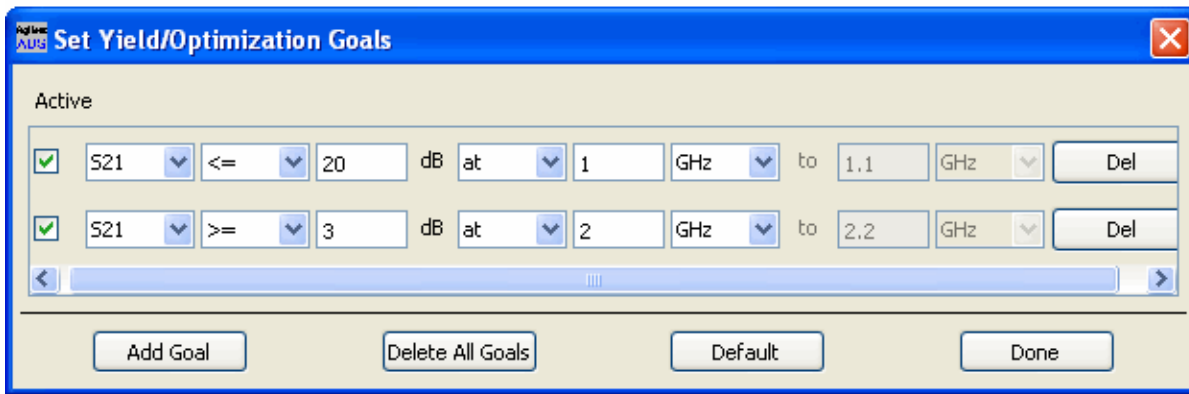
Each goal specifies the insertion loss performance of the network in dB and can represent a specification that the value stay above or below the stated level. The specification can be at a single point, or over a given frequency band.

- *Add Goal* - Add new goals.
- *Del* - Delete individual goals.
- *Delete All Goals* - Delete all goals.
- *Default* - Resets goals to default values.
- *Active* - A goal is 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 are displayed automatically after completion of the analysis.

Starting the Simulation

The yield analysis can accomplished using one of these methods.

- Click the *Simulate* button on the Yield Assistant tab.
- Click the *Simulate Yield* button on the Control window toolbar.
- Choose *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 is 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, you can simulate the SmartComponent manually. To generate a simulation schematic around the selected SmartComponent, click **Create Template** 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, clicking **Close Template** on the Yield Assistant tab returns you to the original design. You can close the simulation schematic manually 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 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.
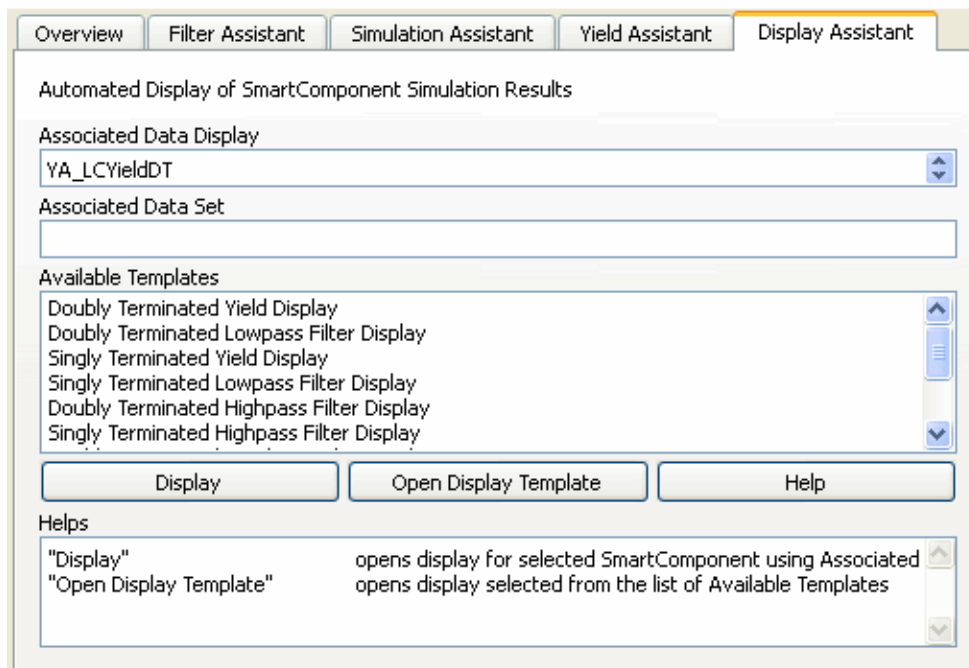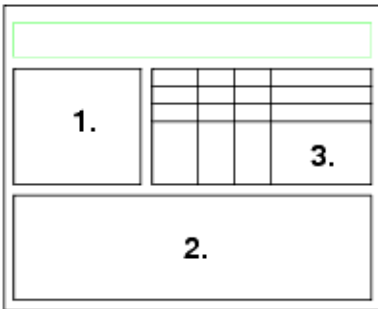
Display Template Features

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 section *SmartComponent Reference* (dgfilter).

Basic Layout

Basic Layout of Display Template shows 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.

Basic Layout of Display Template

Typical Area One Graph

Typical Graphs from Area One shows a typical graph from area one of a display template. The frequency range of the graph is determined by the Simulation Assistant. As you change the frequency range in the Simulation Assistant, this graph updates 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 the marker with the mouse. The performance at the frequency given by M1 is shown in the table in area three.
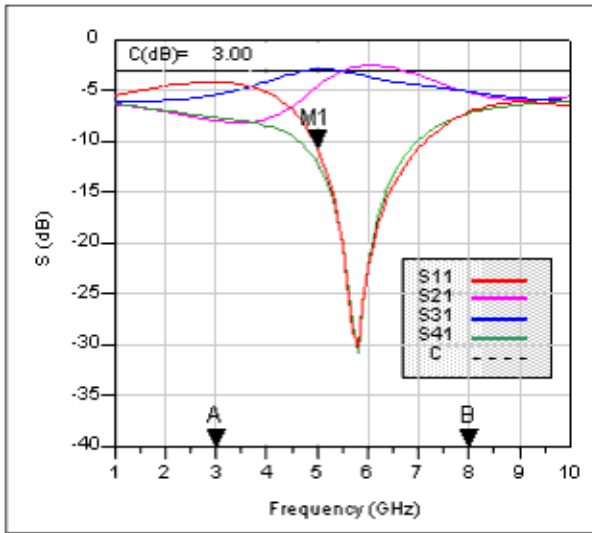
Typical Graphs from Area One

Typical Area Two Graphs

Typical graphs from area 2 of a display template are shown in Typical Graphs from Area Two. 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 is displayed in the table in area three.
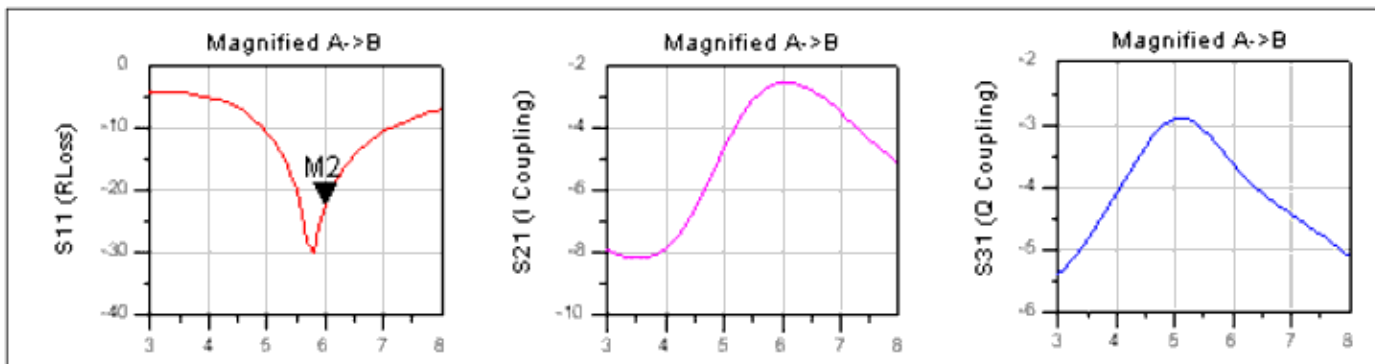
Typical Graphs from Area Two

Typical Area Three Templates

A typical table from area three of a display template is shown in Typical Table from Area Three. The white rows show the 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.

Typical Table from Area Three

Display Assistant Operation

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

Opening a Display

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

- Click the *Display* button on the Display Assistant tab.
- Click the *Display* button on the Control window toolbar.
- Choose *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 is insensitive. If the toolbar or menu are used to try to display the results, a message appears, indicating that no dataset exists.

Display Templates

In some cases, you can use one of the display templates provided with the DesignGuide for other applications. To gain access to one of these templates, select the template from the Available Templates field and click **Open Display Template** on the Control window Display Assistant tab. Insert a dataset of your choice using the dataset pull-down list box in the upper left corner of the display. If you find that some parameters in the display template are not defined in the selected dataset, you can make appropriate modifications to the display. You can save these changes by using the commands in the display **File** menu.

Transformation Assistant

After 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.

The Transformation Assistant is opened 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.
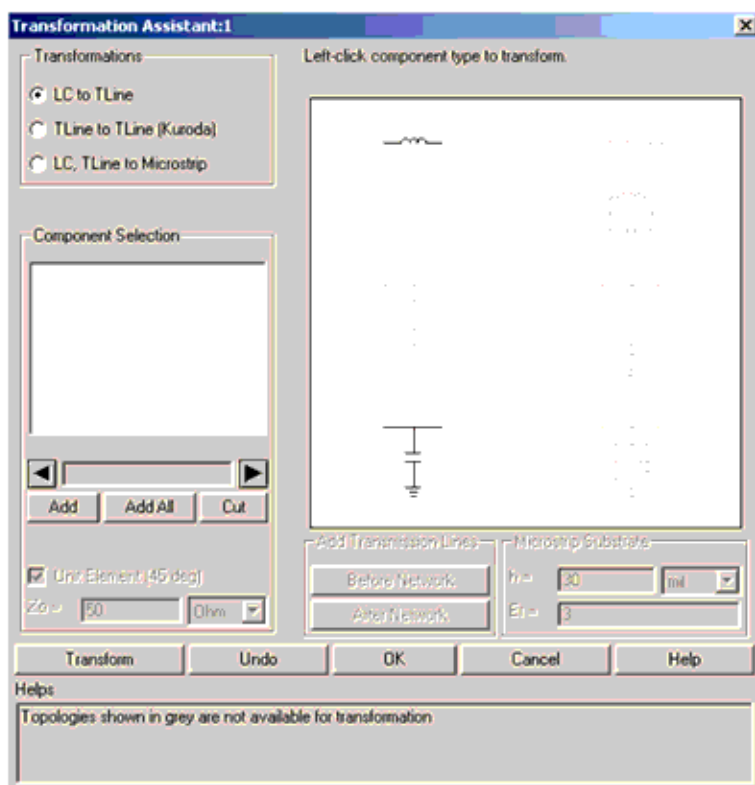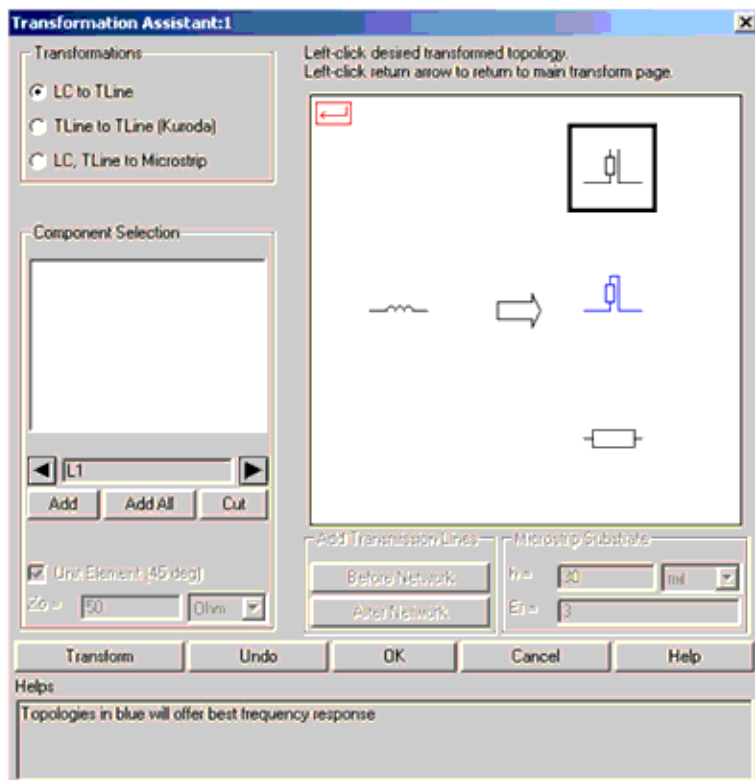
Selecting a Transformation Type

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.

After 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 changes to reveal the different distributed element equivalents available for substitution. Transformations Available for a Series Inductor Circuit 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

After 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 are highlighted, and their instance names (i.e. L1, C4) appears in the text box on the *Transformation Assistant* dialog. The three buttons are used to select the specific components that should be subjected 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. After 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

After all transformations have been made on a specific component type (such as series inductor), performing a left mouse click 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

Five basic transmission line elements can be produced using the Transformation Assistant. These are identified as:

Series Transmission Line

Series Short Circuit Stub

Series Open Circuit Stub

Parallel Short Circuit Stub

Parallel Short Circuit Stub

.
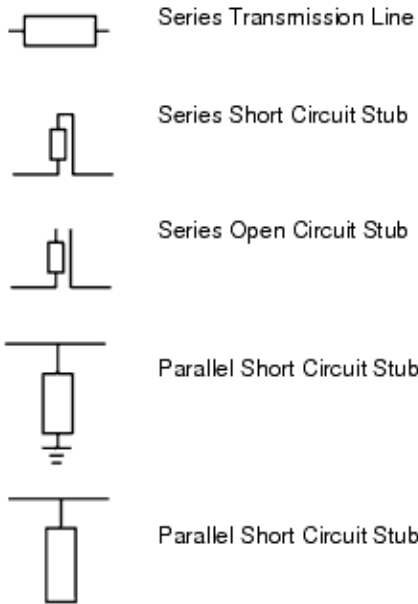
Additional Transformation Functions

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 has an electrical length of 45 degrees and the characteristic impedance is computed appropriately. If the Unit Element box is unchecked, then the Characteristic Impedance (Z0) box becomes active and the computation uses this characteristic impedance to compute the appropriate length.

Characteristic Impedance

The Characteristic Impedance (Z0) box is used to specify the 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 (Z0) 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 (Z0) box. Such lines can be added as needed during the transformation process.

> **ⓘ Note**
> Addition of these lines changes 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 (Er) must be specified. All microstrip elements within a design must share the same substrate parameters. The substrate parameters used in the final design are 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. However, these operations only work on Unit Element lines with electrical lengths of 45 degrees. Therefore, when performing lumped to ideal distributed transformations, you must 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) can be transformed in this way. When adding transmission line elements before or after the network, the electrical length is 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 series stubs 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:
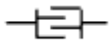
Interdigital Capacitor (can be in series or parallel)
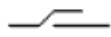
Series Microstrip Thin Film Capacitor

Rectangular Spiral Inductor (can be in series or parallel)

Spiral Inductor (can be in series or parallel)

Parallel Radial Stub

Parallel Butterfly Radial Stub

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 can be required before the substituted device offers the correct performance. In such cases, after 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.

SmartComponent Reference for Filter DesignGuide

This section provides detailed information for all filter SmartComponents.

SmartComponent List

LCLowpassDT (Doubly Terminated Lowpass Filter) (dgfilter)

LCHighpassDT (Doubly Terminated Highpass Filter) (dgfilter)

LCBandpassDT (Doubly Terminated Bandpass Filter) (dgfilter)

LCBandstopDT (Doubly Terminated Bandstop Filter) (dgfilter)

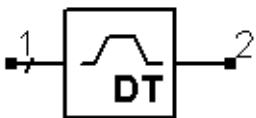*LCLowpassST (Singly Terminated Lowpass Filter)* (dgfilter)

LCHighpassST (Singly Terminated Highpass Filter) (dgfilter)

LCBandpassST (Singly Terminated Bandpass Filter) (dgfilter)

LCBandstopST (Singly Terminated Bandstop Filter) (dgfilter)

LCBandpassDT (Doubly Terminated Bandpass Filter)

Symbol

Parameters

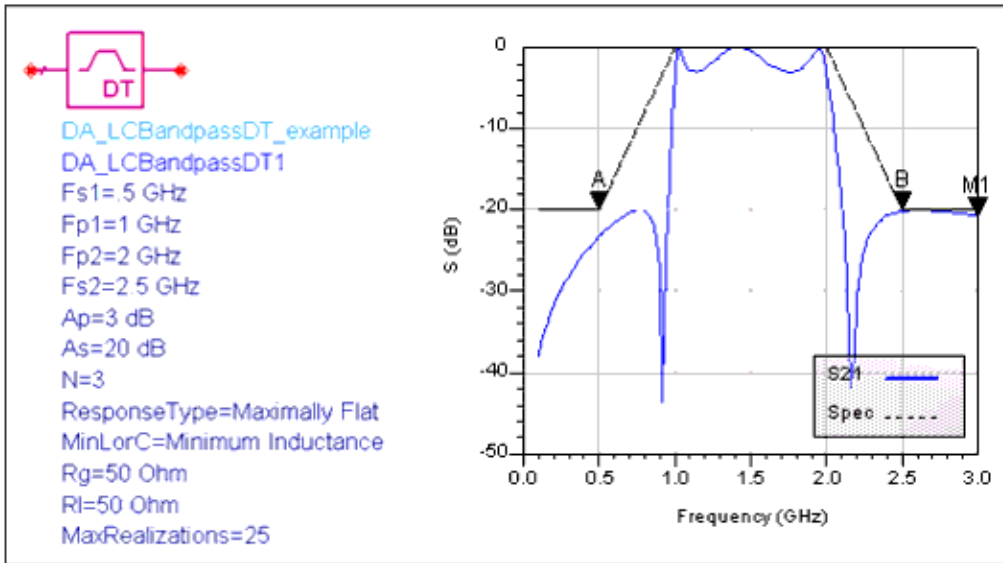| Name	Description	Unit	Default
Fs1	Lower stopband frequency	GHz	.5
Fp1	Lower passband corner frequency	GHz	1
Fp2	Upper passband corner frequency	GHz	2
Fs2	Upper stopband frequency	GHz	2.5
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter, unitless	None	0
ResponseType	Type of filter response	None	Maximally Flat
MinLorC	Minimum inductance or capacitance	None	Minimum Inductance
Rg	Source impedance	Ohm	50
Rl	Load impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

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 (that is, 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 (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.

Figure: Elliptical Bandpass Filter

DA_LCBandpassDT_example
DA_LCBandpassDT1
Fs1=.5 GHz
Fp1=1 GHz
Fp2=2 GHz
Fs2=2.5 GHz
Ap=3 dB
As=20 dB
N=3
ResponseType=Maximally Flat
MinLorC=Minimum Inductance
Rg=50 Ohm
Rl=50 Ohm
MaxRealizations=25

LCBandpassST (Singly Terminated Bandpass Filter)

Symbol

Parameters

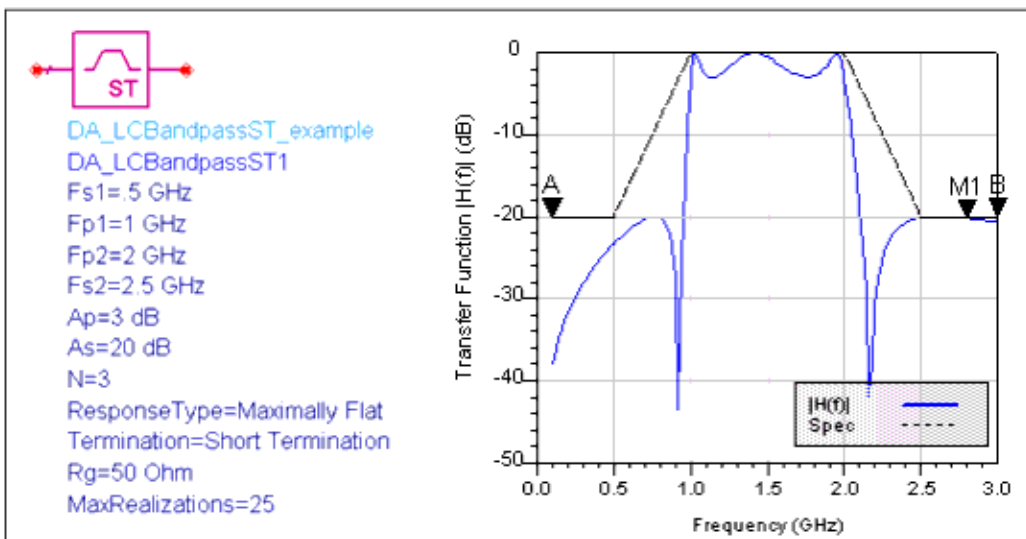| Name	Description	Unit	Default
Fs1	Lower stopband frequency	GHz	.5
Fp1	Lower passband corner frequency	GHz	1
Fp2	Upper passband corner frequency	GHz	2
Fs2	Upper stopband frequency	GHz	2.5
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
Termination	Open or short load termination	None	Short Termination
Rg	Source impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

Notes

1. A singly terminated bandpass filter provides a bandpass frequency response between the input (pin 1) and output (pin 2) ports. This component has a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (that is, 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 (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.

Figure: Elliptical Bandpass Filter

LCBandstopDT (Doubly Terminated Bandstop Filter)

Symbol

Parameters

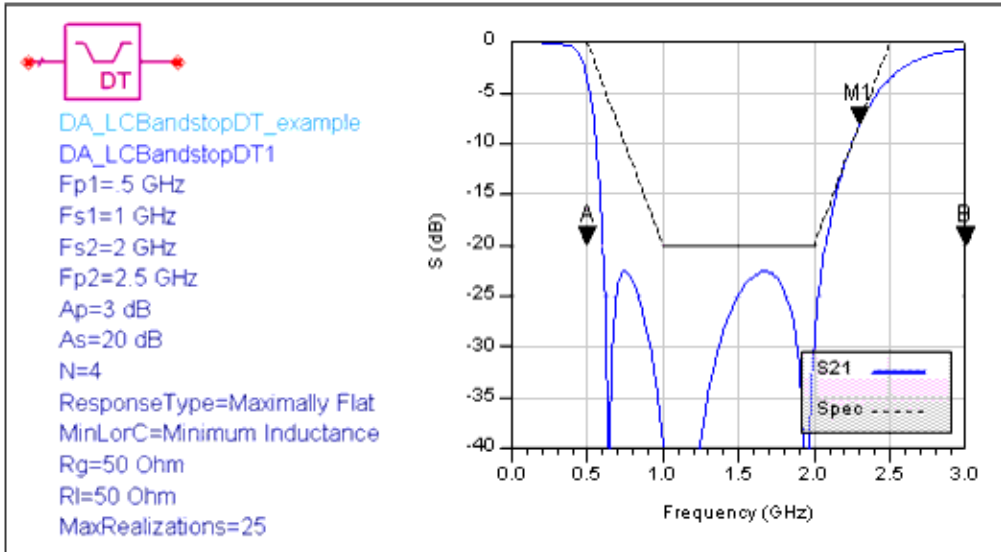| Name	Description	Unit	Default
Fp1	Lower passband corner frequency	GHz	.5
Fs1	Lower stopband frequency	GHz	1
Fs2	Upper stopband frequency	GHz	2
Fp2	Upper passband corner frequency	GHz	2.5
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
MinLorC	Minimum inductance or capacitance	None	Minimum Inductance
Rg	Source impedance	Ohm	50
Rl	Load impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

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 (that is, 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 (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.

Figure: Inverse-Chebyshev Bandstop Filter

DA_LCBandstopDT_example
DA_LCBandstopDT1
Fp1=.5 GHz
Fs1=1 GHz
Fs2=2 GHz
Fp2=2.5 GHz
Ap=3 dB
As=20 dB
N=4
ResponseType=Maximally Flat
MinLorC=Minimum Inductance
Rg=50 Ohm
Rl=50 Ohm
MaxRealizations=25

LCBandstopST (Singly Terminated Bandstop Filter)

Symbol

Parameters

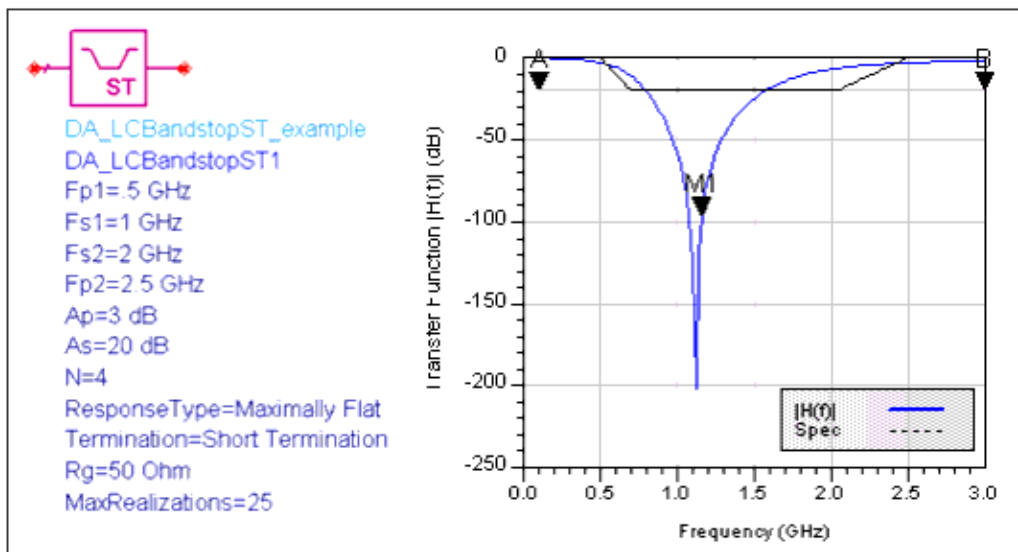| Name	Description	Unit	Default
Fp1	Lower passband corner frequency	GHz	.5
Fs1	Lower stopband frequency	GHz	1
Fs2	Upper stopband frequency	GHz	2
Fp2	Upper passband corner frequency	GHz	2.5
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
Termination	Open or short load termination	None	Short Termination
Rg	Source impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

Notes

1. A singly terminated bandstop filter provides a bandstop frequency response between the input (pin 1) and output (pin 2) ports. This component has a real, non-zero source termination and either an open or short at the load termination.
2. The type of frequency response (that is, 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.
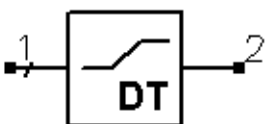
Example

A maximally flat bandstop filter (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.

Figure: Maximally Flat Bandstop Filter

LCHighpassDT (Doubly Terminated Highpass Filter)

Symbol

Parameters

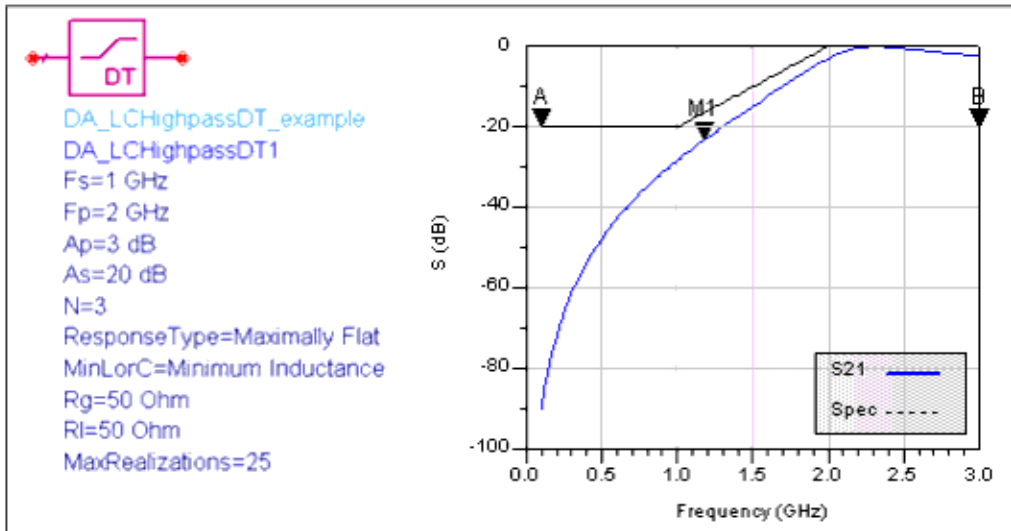| Name	Description	Unit	Default
Fs	Stopband frequency	GHz	1
Fp	Passband corner frequency	GHz	2
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
MinLorC	Minimum inductance or capacitance	None	Minimum Inductance
Rg	Source impedance	Ohm	50
Rl	Load impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

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.
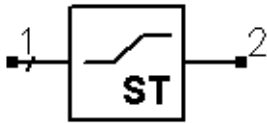
Example

A chebyshev highpass filter (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.

Figure: Chebyshev Highpass Filter

LCHighpassST (Singly Terminated Highpass Filter)

Symbol

Parameters

Name	Description	Unit	Default
Fs	Stopband frequency	GHz	1
Fp	Passband corner frequency	GHz	2
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
Termination	Open or short load termination	None	Short Termination
Rg	Source impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

Notes

1. A singly terminated highpass filter provides a highpass frequency response between

51

the input (pin 1) and output (pin 2) ports. This component has a real, non-zero source termination and either an open or short at the load termination.

2. The type of frequency response (that is, 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.
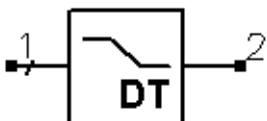
Example

A chebyshev highpass filter (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.

Figure: Chebyshev Highpass Filter
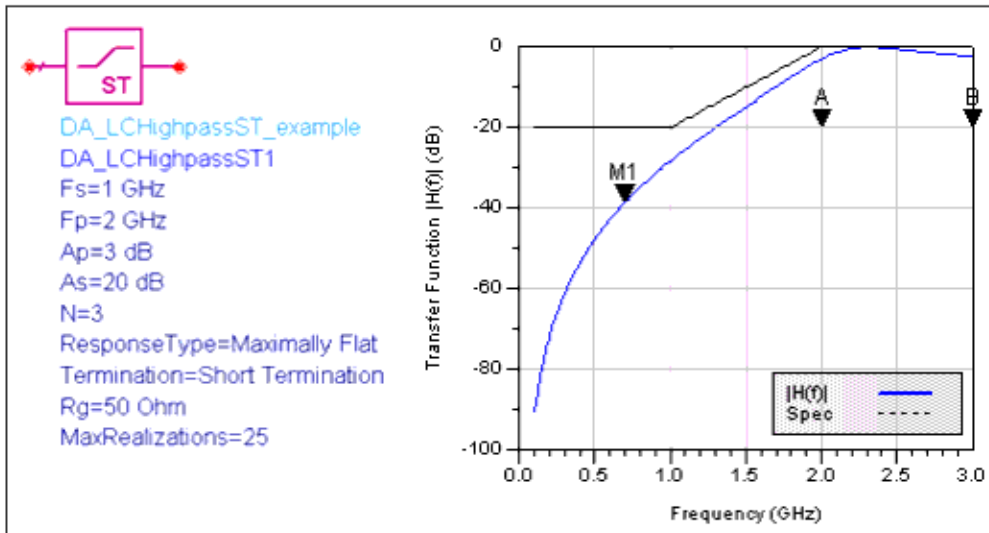
LCLowpassDT (Doubly Terminated Lowpass Filter)

Symbol

Parameters

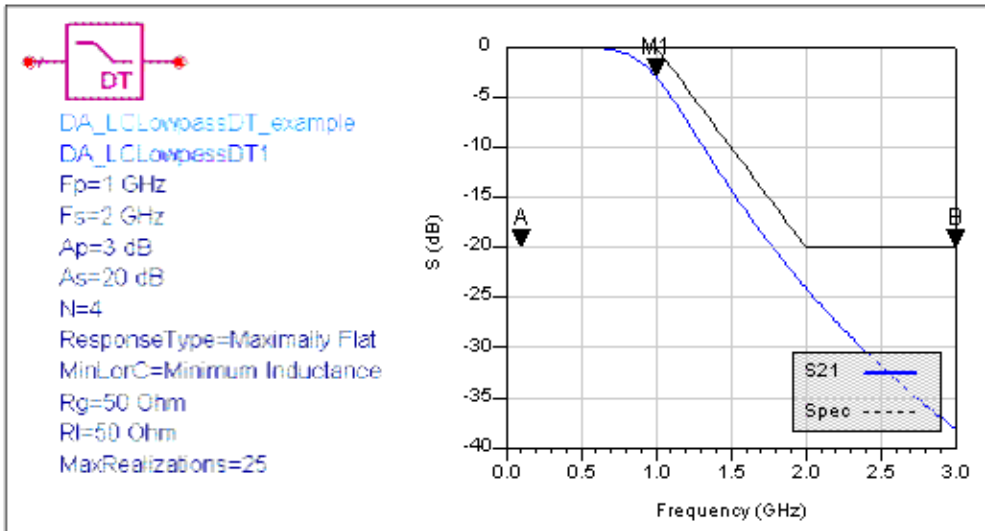| Name	Description	Unit	Default
Fp	Passband corner frequency	GHz	1
Fs	Stopband edge frequency	GHz	2
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
MinLorC	Minimum inductance or capacitance	None	Mininum Inductance
Rg	Source impedance	Ohm	50
Rl	Load impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

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 (that is, 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
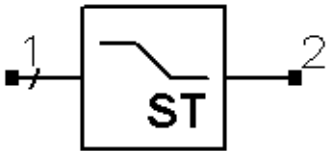
A maximally flat lowpass filter (Maximally Flat Lowpass Filter) was designed for 3dB attenuation at 1 GHz and 20dB attenuation at 2 GHz with matched 50 ohm terminations. This resulted in a filter order of N=4.

Figure: Maximally Flat Lowpass Filter

LCLowpassST (Singly Terminated Lowpass Filter)

Symbol

Parameters

Name	Description	Unit	Default
Fp	Passband corner frequency	GHz	1
Fs	Stopband frequency	GHz	2
Ap	Passband attenuation	dB	3
As	Stopband attenuation	dB	20
N	Order of filter	None	0
ResponseType	Type of filter response	None	Maximally Flat
Termination	Open or short load termination	None	Short Termination
Rg	Source impedance	Ohm	50
MaxRealizations	Maximum number of realizations during synthesis	None	25

Notes

1. A singly terminated lowpass filter provides a lowpass frequency response between the input (pin 1) and output (pin 2) ports. This component has a real, non-zero source termination and either an open or short at the load termination.

54

2. The type of frequency response (that is, 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 (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.

Figure: Maximally Flat Lowpass Filter

DA_LCLowpassST_example
DA_LCLowpassST1
Fp=1 GHz
Fs=2 GHz
Ap=3 dB
As=20 dB
N=4
ResponseType=Maximally Flat
Termination=Short Termination
Rg=50 Ohm
MaxRealizations=25