



Microwave Office Getting Started Guide

ni.com/awr



Microwave Office Getting Started Guide **NI AWR Design Environment**

1960 E. Grand Avenue, Suite 430
El Segundo, CA 90245
USA
Phone: +1 310.726.3000
Fax: +1 310.726.3005
Website: www.ni.com/awr
<awr.support@ni.com>
U.S. Technical Support phone: 888.349.7610

LEGAL NOTICES

© 2019 National Instruments. All rights reserved. © 2019 AWR Corporation. All rights reserved.

Trademarks

- Analog Office, APLAC, AWR, AWR Design Environment, AXIEM, Microwave Office, National Instruments, NI, ni.com and TX-Line are registered trademarks of National Instruments. Visual System Simulator (VSS), Analyst, and AWR Connected are trademarks of AWR Corporation/National Instruments. Refer to the Trademarks section at ni.com/trademarks for other National Instruments trademarks.
- Other product and company names mentioned herein are trademarks or trade names of their respective companies.

Patents

For patents covering NI AWR software products/technology, refer to ni.com/patents.

The information in this guide is believed to be accurate. However, no responsibility or liability is assumed by National Instruments for its use.

Table of Contents

1. Introduction	1-1
Introducing the NI AWR Design Environment	1-1
About This Guide	1-2
Prerequisites	1-2
Contents of this Guide	1-2
Conventions Used in This Guide	1-2
Getting Additional Information	1-3
NI AWR Knowledge Base	1-3
Documentation	1-3
Online Help	1-5
Website Support	1-5
Technical Support	1-5
2. NI AWR Design Environment Suite	2-1
Starting NI AWR Programs	2-2
NI AWR Design Environment Suite Components	2-3
Basic Operations	2-4
Working with Projects	2-4
Project Contents	2-5
Creating, Opening, and Saving Projects	2-5
Opening Example Projects	2-5
Importing Test Benches	2-6
Working with Schematics and Netlists in Microwave Office	2-7
Working with System Diagrams in VSS	2-8
Using the Element Browser	2-9
Adding Subcircuits to Schematics	2-11
Adding Subcircuits to System Diagrams	2-11
Adding Ports to Schematics and System Diagrams	2-11
Connecting Element and System Block Nodes	2-12
Adding Data to Netlists	2-12
Creating EM Structures	2-12
Adding EM Structure Drawings	2-13
Creating a Layout with Microwave Office	2-14
Modifying Layout Attributes and Drawing Properties	2-15
Using the Layout Manager	2-16
Creating Output Graphs and Measurements	2-17
Setting Simulation Frequency and Performing Simulations	2-18
Tuning and Optimizing Simulations	2-19
Using Command Shortcuts	2-19
Using Scripts and Wizards	2-20
Using Online Help	2-20
3. Microwave Office: Importing Data Files	3-1
Importing an S-parameter Data File	3-1
Creating a New Project	3-1
Importing Data Files	3-1
Plotting a Data File Directly	3-2
Adding a Data File to the Schematic	3-4
Creating a Schematic	3-4
Placing a Data File in a Schematic	3-4
Specifying the Simulation Frequency	3-6

Simulating a Schematic with a Data File	3-7
Renormalizing the Data File to a Different Impedance	3-9
4. Microwave Office: Using the Linear Simulator	4-1
Linear Simulations in Microwave Office	4-1
Creating a Lumped Element Filter	4-1
Creating a New Project	4-1
Setting Default Project Units	4-1
Creating a Schematic	4-2
Placing Elements in a Schematic	4-2
Connecting the Wires	4-4
Placing Ports on a Node	4-5
Placing Ground on a Node	4-5
Editing Element Parameters	4-6
Specifying the Simulation Frequency	4-6
Creating a Graph	4-7
Adding a Measurement	4-8
Analyzing the Circuit	4-9
Adding Auto-Search Markers	4-9
Tuning the Circuit	4-11
Creating Variables	4-13
Adding Optimization Goals	4-14
Optimizing the Circuit	4-15
5. Microwave Office: Creating Layouts from Schematics	5-1
Layouts in Microwave Office	5-1
Layout Tips and Tricks	5-1
Creating a Layout from a Schematic	5-1
Creating a New Project	5-2
Importing a Layer Process File	5-2
Editing Database Units and Default Grid Size	5-3
Importing a GDSII Cell Library	5-4
Importing a Data File	5-4
Placing a Data File in a Schematic and Adding a Ground Node	5-4
Changing the Element Symbol	5-5
Placing Microstrip Elements for Layout	5-6
Assigning an Artwork Cell to a Schematic Element	5-9
Viewing a Layout	5-9
Snapping Layout	5-10
Running the Connectivity Checker	5-10
Anchoring a Layout Cell	5-12
Creating an Artwork Cell	5-13
Add Ports to an Artwork Cell	5-16
Editing the Schematic and Assigning a Chip Cap Cell	5-17
Routing the MTRACE2 Element in Layout	5-19
Snapping Functions for Layout Cells	5-21
Exporting the Layout	5-25
6. Microwave Office: Using the Nonlinear Simulator	6-1
Harmonic Balance in Microwave Office	6-1
Single-Tone Analysis	6-1
Multi-Tone Analysis	6-1
Nonlinear Measurements	6-1
Creating a Power Amplifier Circuit	6-1

Creating a New Project	6-2
Setting Default Project Units	6-2
Creating a Schematic	6-3
Placing a Nonlinear Model from the Library	6-3
Placing an IV Curve Meter on the Nonlinear Element	6-4
Editing the IV Curve Meter Element	6-5
Adding an IV Curve Measurement	6-5
Creating a Bias Circuit	6-6
Adding Schematic Back Annotation	6-8
Adding a Harmonic Balance Port	6-10
Specifying Nonlinear Simulation Frequencies	6-12
Adding a Large Signal Reflection Coefficient Measurement	6-13
Importing Input Match and Output Match Schematics	6-14
Adding Subcircuits to a Schematic	6-16
Creating a Pout vs. Frequency Measurement	6-19
Creating a Dynamic Load Line Measurement	6-21
Setting up a Two-Tone Simulation	6-23
Copying a Schematic in the Project Browse	6-23
Adding a Two-Tone Harmonic Balance Port	6-23
Adding a Third-Order Intermodulation Measurement	6-24
Using Variable Sweeps to Measure IP3 vs Voltage	6-25
Plotting IM3 vs Output Power	6-27
7. Microwave Office: Using the Electromagnetic Simulator	7-1
EM Simulation in Microwave Office	7-1
Creating a Distributed Interdigital Filter	7-2
Creating a New Project	7-2
Importing a Layer Process File (LPF)	7-2
Creating an EM Structure	7-3
Setting Up the Enclosure	7-4
Sidewall Boundary Conditions	7-8
Adding Conductors to the Layout	7-9
Adding Vias	7-15
Viewing the Structure in 3D	7-16
Adding Ports	7-17
Specifying the Simulation Frequencies	7-18
Previewing the Geometry	7-20
Viewing Structure Mesh	7-22
Running the EM Simulator	7-23
Displaying Results on a Graph	7-25
Changing Frequency Range and Step Size	7-26
Animating Currents	7-27
Completing the Filter Layout	7-29
Advanced Frequency Sweep	7-34
Adding an EM Structure to a Schematic as a Subcircuit	7-37
Index	Index-1

Chapter 1. Introduction

The following NI AWR Design Environment Getting Started Guides are available:

- The *Microwave Office Getting Started Guide* provides step-by-step examples that show you how to use Microwave Office software to create circuit designs.
- The *Analyst Getting Started Guide* provides step-by-step examples that show you how to use Analyst™ software to create and simulate 3D EM structures from the Microwave Office program.
- *MMIC Getting Started Guide* provides step-by-step examples that show you Monolithic Microwave Integrated Circuit (MMIC) features and designs.
- *Visual System Simulator Getting Started Guide* provides step-by-step examples that show you how to use Visual System Simulator™ (VSS) software to create system simulations and to incorporate Microwave Office circuit designs.
- *Analog Office Getting Started Guide* provides step-by-step examples that show you how to use Analog Office to create circuit designs and display various measurements in graphical form.

To set up the NI AWR Design Environment software for PCB style design, choose **Tools > Create New Process** to display the Create New Process dialog box, then click the **Help** button for details on using this tool.

Introducing the NI AWR Design Environment

Welcome to the NI AWR Design Environment suite!

This suite comprises three powerful tools that can be used together to create an integrated system, RF, or analog design environment: Visual System Simulator (VSS), Microwave Office, and Analog Office software. These powerful tools are fully integrated in the NI AWR Design Environment suite and allow you to incorporate circuit designs into system designs without leaving the design environment.

VSS software enables you to design and analyze end-to-end communication systems. You can design systems composed of modulated signals, encoding schemes, channel blocks and system level performance measurements. You can perform simulations using VSS software's predefined transmitters and receivers, or you can build customized transmitters and receivers from basic blocks. Based on your analysis needs, you can display BER curves, ACPR measurements, constellations, and power spectrums, to name a few. VSS software provides a real-time tuner that allows you to tune the designs and then see your changes immediately in the data display.

Microwave Office and Analog Office software enable you to design circuits composed of schematics and electromagnetic (EM) structures from an extensive electrical model database, and then generate layout representations of these designs. You can perform simulations using any of NI AWR's simulation engines, such as a linear simulator, an advanced harmonic balance simulator for nonlinear frequency-domain simulation and analysis (the APLAC harmonic balance simulator), a 3D-planar EM simulator (the AXIEM tool), a 3D-FEM simulator (the Analyst tool), transient circuit simulators (the APLAC transient simulator or an optional HSPICE simulator) -- and display the output in a wide variety of graphical forms based on your analysis needs. You can then tune or optimize the designs and your changes are automatically and immediately reflected in the layout. Statistical analysis allows you to analyze responses based on statistically varying design components.

Analog Office provides a single environment to fully interact with a comprehensive and powerful set of integrated tools for top-down and front-to-back analog and RFIC design.

The tool set spans the entire IC design flow, from system-level to circuit-level design and verification, including design entry and schematic capture, time- and frequency-domain simulation and analysis, physical layout with automated device-level place and route and integrated design rule checker (DRC), 3D full field solver-based extraction with industry

gold standard high-speed extraction technology from OEA International, and a comprehensive set of waveform display and analysis capabilities supporting complex RF measurements.

OBJECT ORIENTED TECHNOLOGY

At the core of the NI AWR Design Environment software capability is advanced object-oriented technology. This technology results in software that is compact, fast, reliable, and easily enhanced with new technology as it becomes available.

About This Guide

This Getting Started Guide is designed to familiarize you with the software by demonstrating Microwave Office, VSS, Analog Office, Analyst, or MMIC capabilities through working examples.

Prerequisites

You should be familiar with Microsoft® Windows® and have a working knowledge of basic circuit and/or system design and analysis.

This document is available as a PDF file on your Program Disk (*_Getting_Started.pdf, depending on your product), or you can download it from the Knowledge Base at [NI AWR Knowledge Base](#).

If you are viewing this guide as online Help and intend to work through the examples, you can obtain and print out the PDF version for ease of use.

Contents of this Guide

Chapter 2 provides an overview of the NI AWR Design Environment suite including the basic menus, windows, components and commands.

In the *Microwave Office Getting Started Guide* the subsequent chapters take you through hands-on examples that show you how to use Microwave Office software to create circuit designs including layout and AXIEM 3D planar EM layout and simulation.

In the *Visual System Simulator Getting Started Guide* the subsequent chapters take you through hands-on examples that show you how to use VSS software to create system simulations and to incorporate Microwave Office circuit designs.

In the *Analog Office Getting Started Guide* the subsequent chapters take you through hands-on examples that show you how to use Analog Office to create circuit designs and display various measurements in graphical form.

In the *MMIC Getting Started Guide* the subsequent chapters take you through hands-on examples that show you Monolithic Microwave Integrated Circuit (MMIC) features and designs.

In the *Analyst Getting Started Guide* the subsequent chapters take you through hands-on examples that show use of the Analyst 3D Electromagnetic simulator for 3D EM simulation within Microwave Office software. Use of 3D parametric layout cells and a 3D Layout Editor is included.

Conventions Used in This Guide

This guide uses the following typographical conventions:

Item	Convention
Anything that you select (or click) in the program, like menu items, button names, and dialog box option names	Shown in a bold alternate type. Nested menu selections are shown with a ">" to indicate that you select the first menu item and then select the second menu item: Choose File > New Project
Any text that you enter using the keyboard	Shown in a bold type within quotation marks: Enter " my_project " in Project Name .
File names and directory paths.	Shown in italics: <i>C:\Program Files\AWR\AWRDE\14</i> or <i>C:\Program Files (x86)\AWR\AWRDE\14</i> is the default installation directory.
Keys or key combinations that you press	Shown in a bold alternate type with initial capitals. Key combinations are shown with a "+" to indicate that you press and hold the first key while pressing the second key: Press Alt + F1 .

Getting Additional Information

There are multiple resources available for additional information and technical support for NI AWR products.

NI AWR Knowledge Base

www.awrcorp.com/support/help.aspx?id=9

- Application Notes - Technical papers on various topics written by NI AWR or our partners.
- Examples - Pages explaining examples available both in the installed software and those only available for download.
- Licensing - Step by step guide to resolving most licensing problems.
- Questions - Questions and answers for common customer issues.
- Scripts - Scripted utilities to help solve specific problems.
- Documentation - Complete copy of the latest released documentation.
- Videos - Short technical videos on how to accomplish specific tasks.

Documentation

Documentation for the NI AWR Design Environment suite includes:

- *What's New in NI AWR Design Environment 14?* presents the new or enhanced features, elements, system blocks, and measurements for this release. This document is available in the Help by clicking the Windows **Start** button and choosing **All Programs > AWRDE 14 > AWR Design Environment Help** and then expanding the **NI AWR Design Environment** node on the **Contents** tab, or by choosing **Help > What's New** while in the program.
- The *Installation Guide* describes how to install the NI AWR Design Environment suite and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips. This document is available on your Program Disk as *install.pdf*, or downloadable from the Knowledge Base at [NI AWR Knowledge Base](#).

- The [User Guide](#) provides an overview of the NI AWR Design Environment suite including chapters on the user interface; using schematics/system diagrams; data files; netlists; graphs, measurements, and output files; and variables and equations in projects. In addition, an appendix providing guidelines for starting a new design is included.
- The [Simulation and Analysis Guide](#) discusses simulation basics such as swept parameter analysis, tuning/optimizing/yield, and simulation filters; and provides simulation details for DC, linear, AC, harmonic balance, transient, and EM simulation/extraction theory and methods.
- The [Dialog Box Reference](#) provides a comprehensive reference of all program dialog boxes with dialog box graphics, overviews, option details, and information on how to navigate to each dialog box.
- The [API Scripting Guide](#) explains the basic concepts of NI AWR Design Environment scripting and provides coding examples. It also provides information on the most useful objects, properties, and methods for creating scripts in the NI AWR Script Development Environment (NI AWR SDE). In addition, this guide contains the NI AWR Design Environment Component API list.
- The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. This document is available within the program by choosing **Help > Quick Reference** or on your Program Disk as *Quick_Reference.pdf*. This is an excellent document to print and keep handy at your desk.
- Context sensitive Help is available for most operations or phases of design creation. To view an associated Help topic, press the **F1** key during design creation.

Documentation for Microwave Office and Analog Office software includes:

- The [Microwave Office Layout Guide](#) contains information on creating and viewing layouts for schematics and EM structures, including use of the Layout Manager, Layout Process File, artwork cell creation/editing/properties, Design Rule Checking, and other topics.
- The [Microwave Office Element Catalog](#) provides complete reference information on all of the electrical elements that you use to build schematics.
- The [Microwave Office Measurement Catalog](#) provides complete reference information on the "measurements" (for example, computed data such as gain, noise, power, or voltage) that you can choose as output for your simulations.

Documentation for VSS software includes:

- The [VSS System Block Catalog](#) provides complete reference information on all of the system blocks that you use to build systems.
- The [VSS Measurement Catalog](#) provides complete reference information on the measurements you can choose as output for your simulations.
- The [VSS Modeling Guide](#) contains information on simulation basics, RF modeling capabilities, and noise modeling.

Documentation for the 3D Editor and Analyst-MP product(stand-alone product for multi-physics types of EM problems) includes:

- The *What's New in Analyst-MP 14 (Analyst_Whats_New.pdf)* presents the new or enhanced features for both the 3D Layout Editor and Analyst-MP software.
- The *Analyst Installation Guide (Analyst_Install.pdf)* describes how to install Analyst software and configure it for locked or floating licensing options. It also provides licensing configuration troubleshooting tips.
- The *Analyst-MP Getting Started Guide (Analyst_Getting_Started.pdf)* provides step-by-step examples that show you how to use Analyst-MP software.

- The *Analyst User Guide (Analyst_User_Guide.pdf)* provides an overview of the 3D Editor and Analyst-MP software; including chapters on the user interface, structures, simulations, post processing, variables, data files, and scripting.

Online Help

All NI AWR Design Environment documentation is available as on-line Help.

To access online Help, choose **Help** from the menu bar or press **F1** anywhere in the program. Context sensitive help is available for elements and system blocks in the Element Browser and within schematics or system diagrams. Context sensitive Help is available for measurements from the Add/Modify Measurement dialog box.

Website Support

Support is also available from the NI AWR website at www.ni.com/awr. You can go directly to this site from the NI AWR Design Environment suite **Help** menu. The Support page provides links to the following:

- the current software version
- the Knowledge Base, which contains Frequently Asked Questions (FAQs) from users, Application Notes, Tutorials, and project examples
- All Microwave Office, VSS, and Analog Office documentation

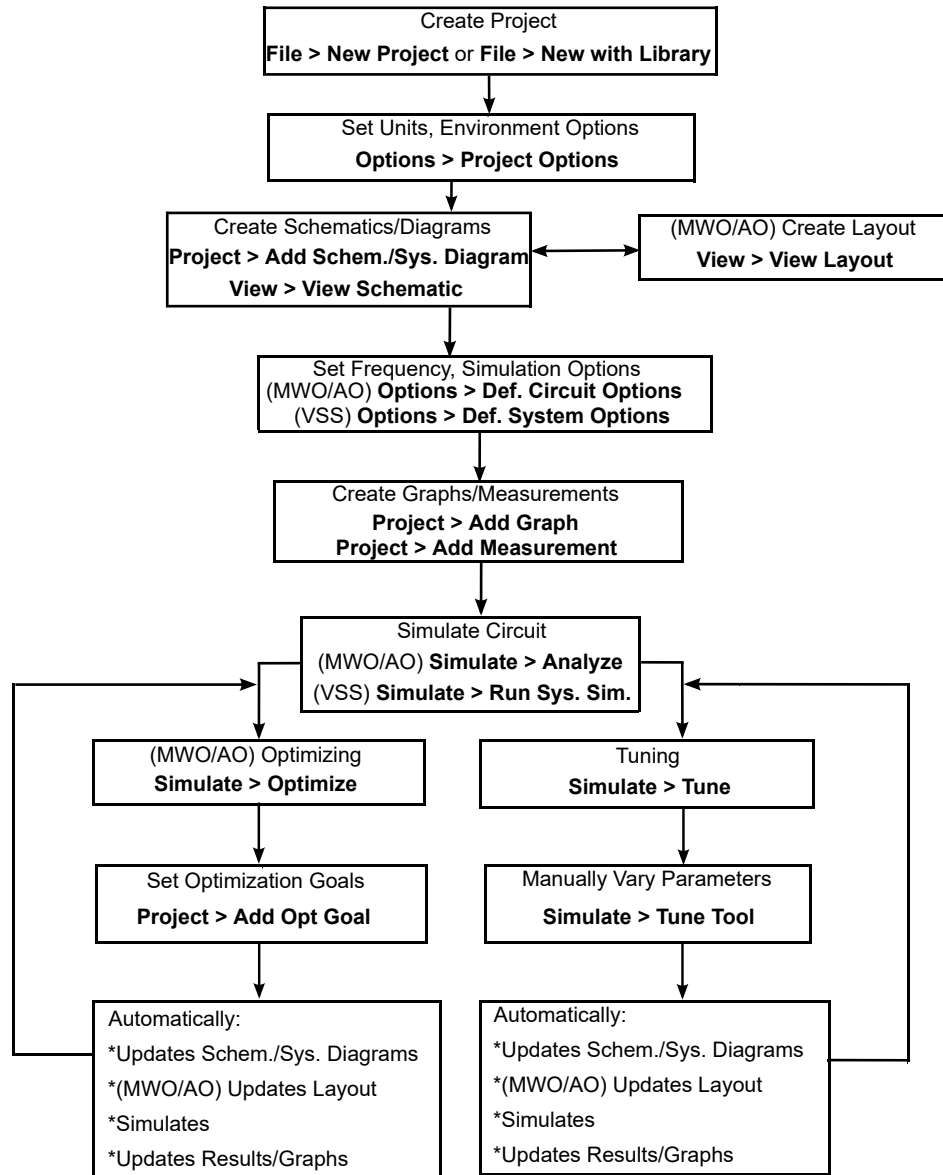
Technical Support

Technical Support is available Monday - Friday, 7 a.m. - 5 p.m., PST.

Phone: 888.349.7610 / Fax: 310.726.3005 / E-mail: <awr.support@ni.com>.

Chapter 2. NI AWR Design Environment Suite

The basic design flow in the NI AWR Design Environment suite is shown in the following flow chart.



This chapter describes the windows, menus and basic operations for performing the following tasks in the NI AWR Design Environment suite:

- Creating projects to organize and save your designs
- Creating system diagrams, circuit schematics, and EM structures
- Placing circuit elements into schematics
- Placing system blocks into system diagrams

- Incorporating subcircuits into system diagrams and schematics
- Creating layouts
- Creating and displaying output graphs
- Running simulations for schematics and system diagrams
- Tuning simulations

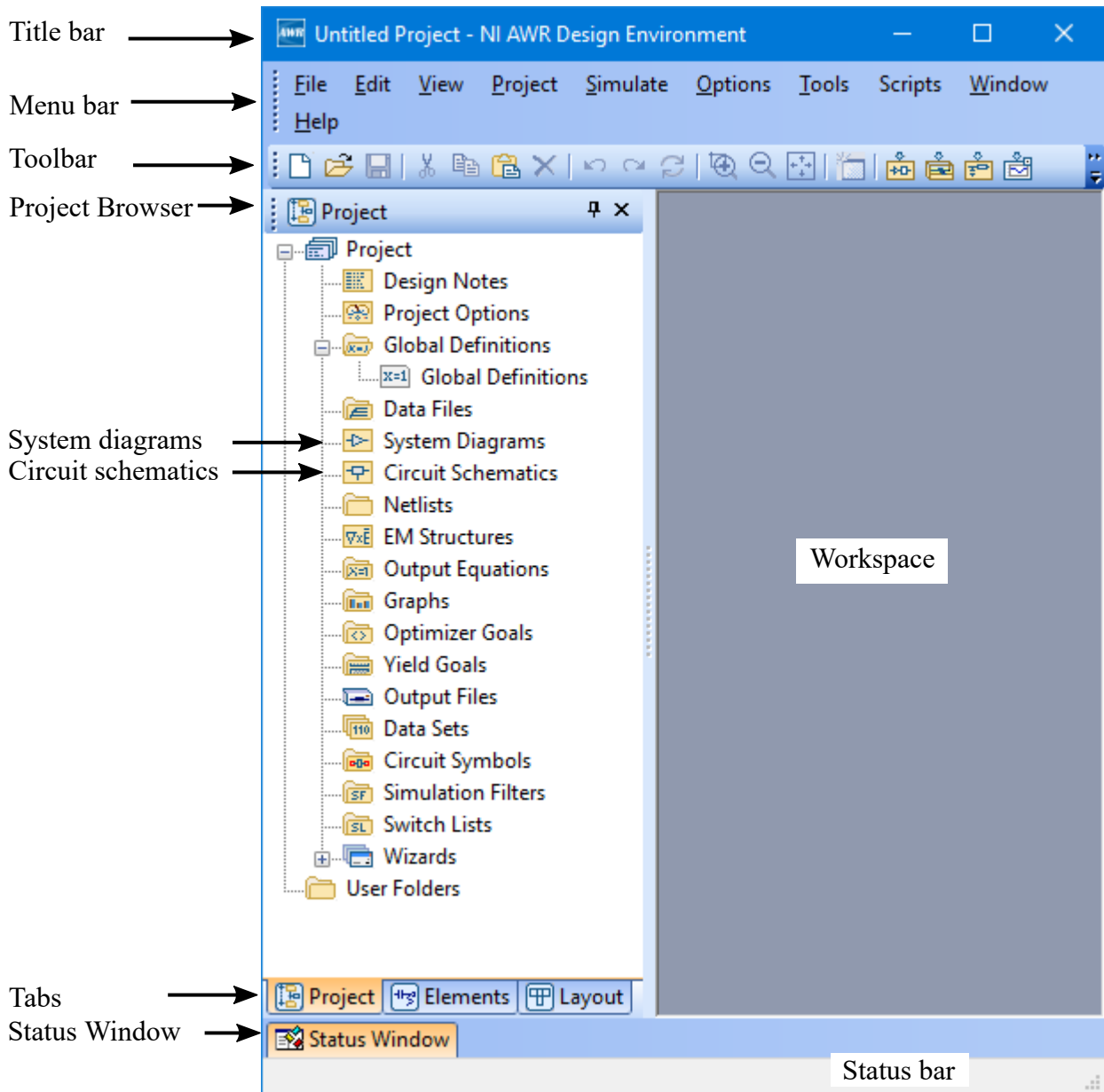
NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Starting NI AWR Programs

To start the NI AWR Design Environment suite:

1. Click the Windows **Start** button.
2. Choose **All Programs > AWRDE 14 > AWR Design Environment 14**.

The following main window displays.



If the NI AWR Design Environment suite was not configured during installation to display in your **Start** menu, start the application by double-clicking the **My Computer** icon on your desktop, opening the drive and folder where you installed the program, and double-clicking on *MWOoffice.exe*, the NI AWR Design Environment application.

NI AWR Design Environment Suite Components

The NI AWR Design Environment suite contains the windows, components, menu selections and tools you need to create linear and nonlinear schematics, set up EM structures, generate circuit layouts, create system diagrams, perform simulations, and display graphs. Most of the basic procedures apply to Microwave Office, Visual System Simulator™ (VSS), and Analog Office software. The major components of the NI AWR Design Environment suite are:

Component	Description
Title bar	The title bar displays the name of the open project and any Process Design Kit (PDK) used with the project.
Menu bar	The menu bar comprises the set of menus located along the top of the window for performing a variety of Microwave Office, VSS, and Analog Office tasks.
Toolbar	The toolbar is the row of buttons located just below the menu bar that provides shortcuts to frequently used commands such as creating new schematics, performing simulations, or tuning parameter values or variables. The buttons available depend on the functions in use and the active window within the design environment (as well as any customization of toolbar button groups). Position the cursor over a button to view the button name/function.
Workspace	The workspace is the area in which you design schematics and diagrams, draw EM structures, view and edit layouts, and view graphs. You can use the scrollbars to move around the workspace. You can also use the zoom in and zoom out options from the View menu.
Project Browser (Project tab)	Located by default in the left column of the window, this is the complete collection of data and components that define the currently active project. Items are organized into a tree-like structure of nodes and include schematics, system diagrams and EM structures, simulation frequency settings, output graphs, user folders and more. The Project Browser is active when the NI AWR Design Environment suite first opens, or when you click the Project tab. Right-click a node in the Project Browser to access menus of relevant commands.
Element Browser (Elements tab)	The Element Browser contains a comprehensive inventory of circuit elements for building your schematics, and system blocks for building system diagrams for simulations. The Element Browser displays by default in the left column in place of the Project Browser when you click the Elements tab.
Layout Manager (Layout tab)	The Layout Manager contains options for viewing and drawing layout representations, creating new layout cells, and working with artwork cell libraries. The Layout Manager displays by default in the left column in place of the Project Browser when you click the Layout tab.
Status Window (Status Window tab)	The Status Window displays error, warning, and informational messages about the current operation or simulation. The Status Window displays by default at the bottom of the workspace when you click the Status Window tab.
Status bar	The bar along the very bottom of the design environment window that displays information dependent on what is highlighted. For example, when an element in a schematic is selected, the element name and ID displays. When a polygon is selected, layer and size information displays, and when a trace on a graph is selected, the value of a swept parameter displays.

You can invoke many of the functions and commands from the menus and on the toolbar, and in some cases by right-clicking a node in the Project Browser. This guide may not describe all of the ways to invoke a specific task.

Basic Operations

This section highlights the windows, menu choices, and commands available for creating simulation designs and projects in the NI AWR Design Environment suite. Detailed use information is provided in the chapters that follow.

Working with Projects

The first step in building and simulating a design is to create a project. You use a project to organize and manage your designs and everything associated with them in a tree-like structure.

Project Contents

Because Microwave Office, VSS, and Analog Office programs are fully integrated in the NI AWR Design Environment suite, you can start a project based on a system design using VSS software, or on a circuit design using Microwave Office or Analog Office software. The project may ultimately combine all elements. You can view all of the components and elements in the project in the Project Browser. Modifications are automatically reflected in the relevant elements.

A project can include any set of designs and one or more linear schematics, nonlinear schematics, EM structures, or system level blocks. A project can include anything associated with the designs, such as global parameter values, imported files, layout views, and output graphs.

Creating, Opening, and Saving Projects

When you first start the NI AWR Design Environment suite, a default empty project titled "Untitled Project" is loaded. Only one project can be active at a time. The name of the active project displays in the main window title bar.

After you create (name) a project, you can create your designs. You can perform simulations to analyze the designs and see the results on a variety of graphical forms. Then, you can tune or optimize parameter values and variables as needed to achieve the desired response. You can generate layout representations of the designs, and output the layout to a DXF, GDSII, or Gerber file. See [Appendix B, *New Design Considerations*](#) in *User Guide* of the *User Guide* for advanced guidelines on starting a new design.

To create a project choose **File > New Project**. Name the new project and the directory you want to write it to by choosing **File > Save Project As**. The project name displays in the title bar.

To open an existing project, choose **File > Open Project**. To save the current project, choose **File > Save Project**. When you save a project, everything associated with it is automatically saved. NI AWR projects are saved as *.emp files.

Opening Example Projects

NI AWR provides a number of project examples (*.emp files) in the installation directory to demonstrate key concepts, program functions and features, and show use of specific elements.

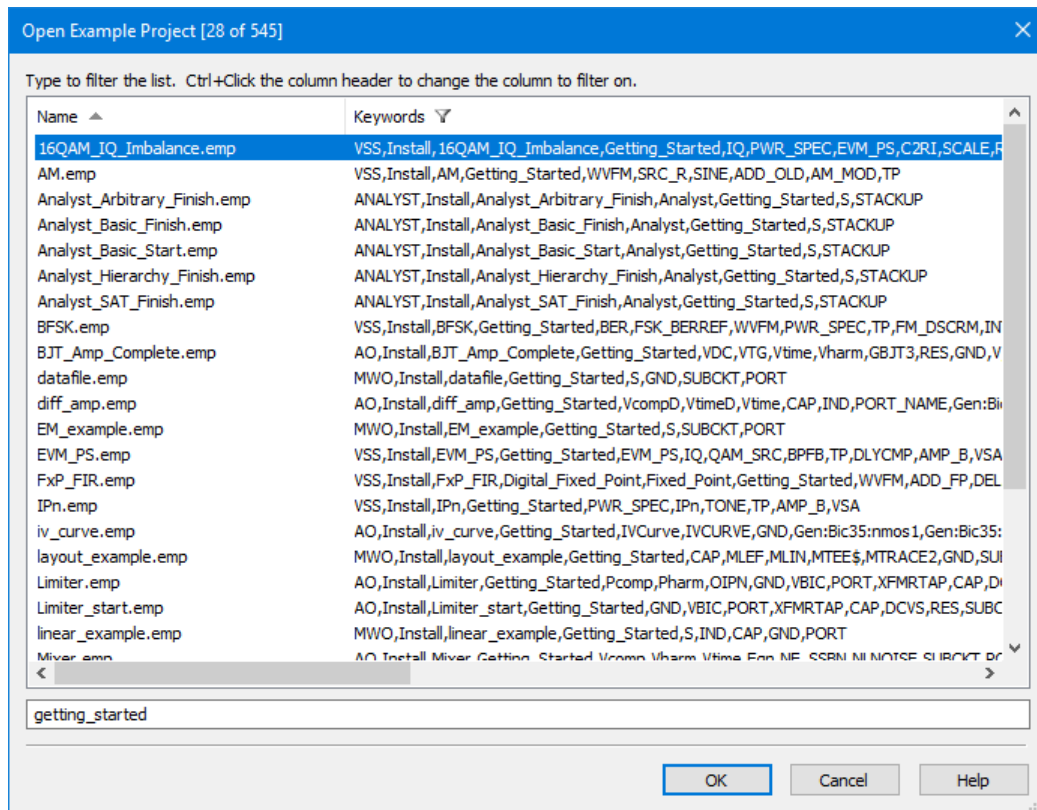
To search for and open example projects referenced in this guide:

1. Choose **File > Open Example**.

The Open Example Project dialog box displays with columns for the project name and keywords associated with each example project.

2. Filter the list using "getting_started" as a keyword by **Ctrl**-clicking the **Keywords** column header and typing "**getting_started**" in the text box at the bottom of the dialog box.

As shown in the following figure, the example list is filtered to display only those projects that have the "getting_started" keyword associated with them.



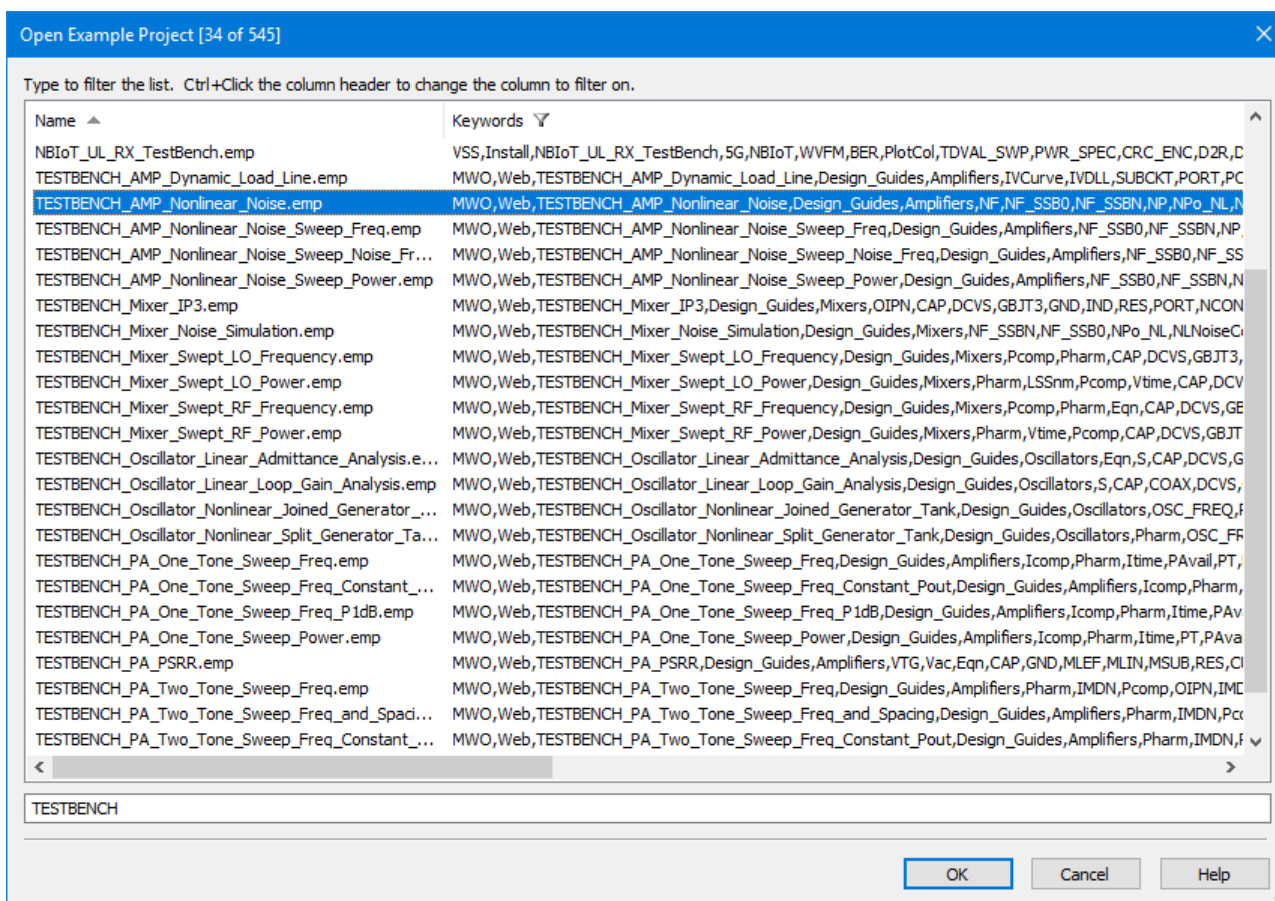
NOTE: You can filter examples by keyword or by file name. An inverted triangle in the column header indicates the column on which your search is filtered. Press the **Ctrl** key while clicking a column header to change which column is used to filter.

Importing Test Benches

NI AWR provides several test bench examples that can serve as design guides for various applications such as mixers, amplifiers, and oscillators. These test benches are set up for import into your working project.

To import a test bench into your project:

1. Choose **File > Import Project**.
2. Browse to *C:\Program Files\AWR\AWRDE\14\Examples* or *C:\Program Files (x86)\AWR\AWRDE\14\Examples* and import the desired test bench. The test bench project file names are prefaced with "TESTBENCH" as shown in the following figure.



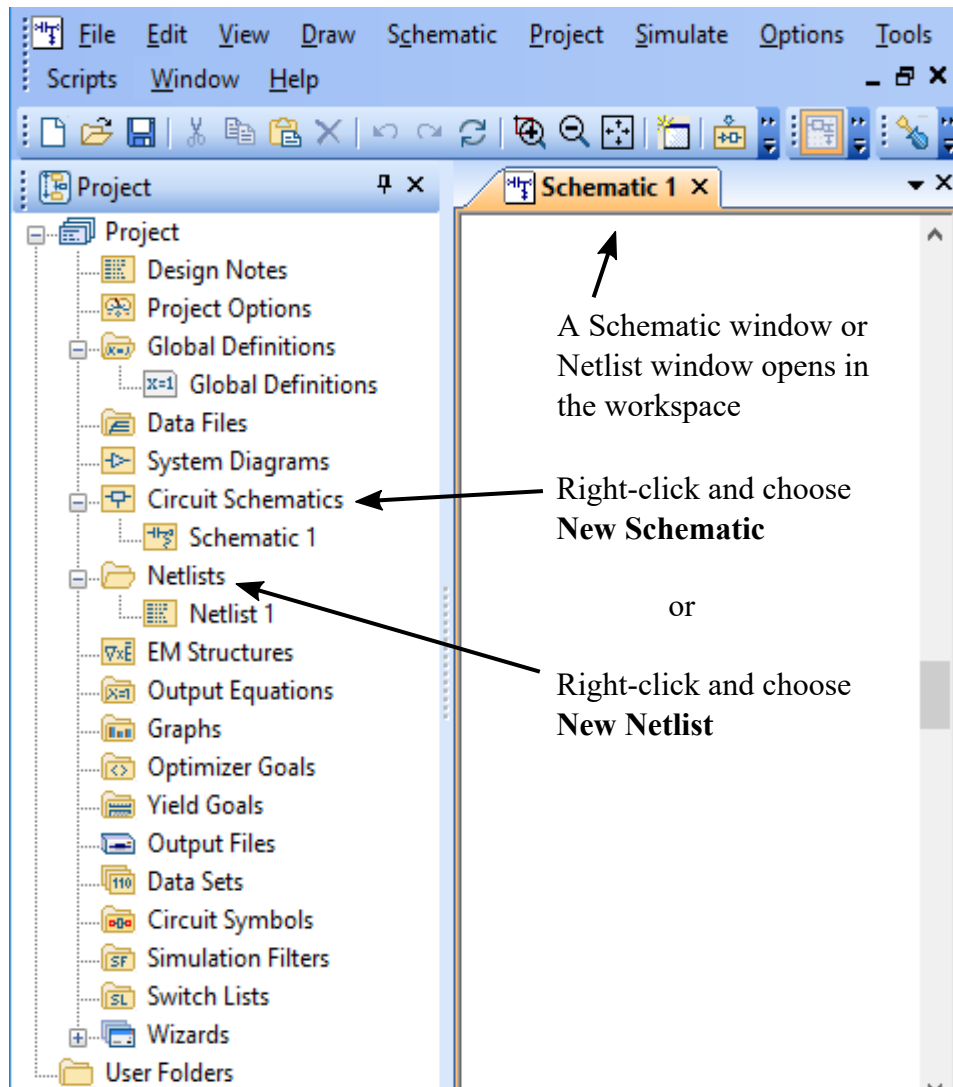
Working with Schematics and Netlists in Microwave Office

A schematic is a graphical representation of a circuit, while a netlist is a text-based description.

To create a schematic, right-click **Circuit Schematics** in the Project Browser, choose **New Schematic**, and then specify a schematic name.

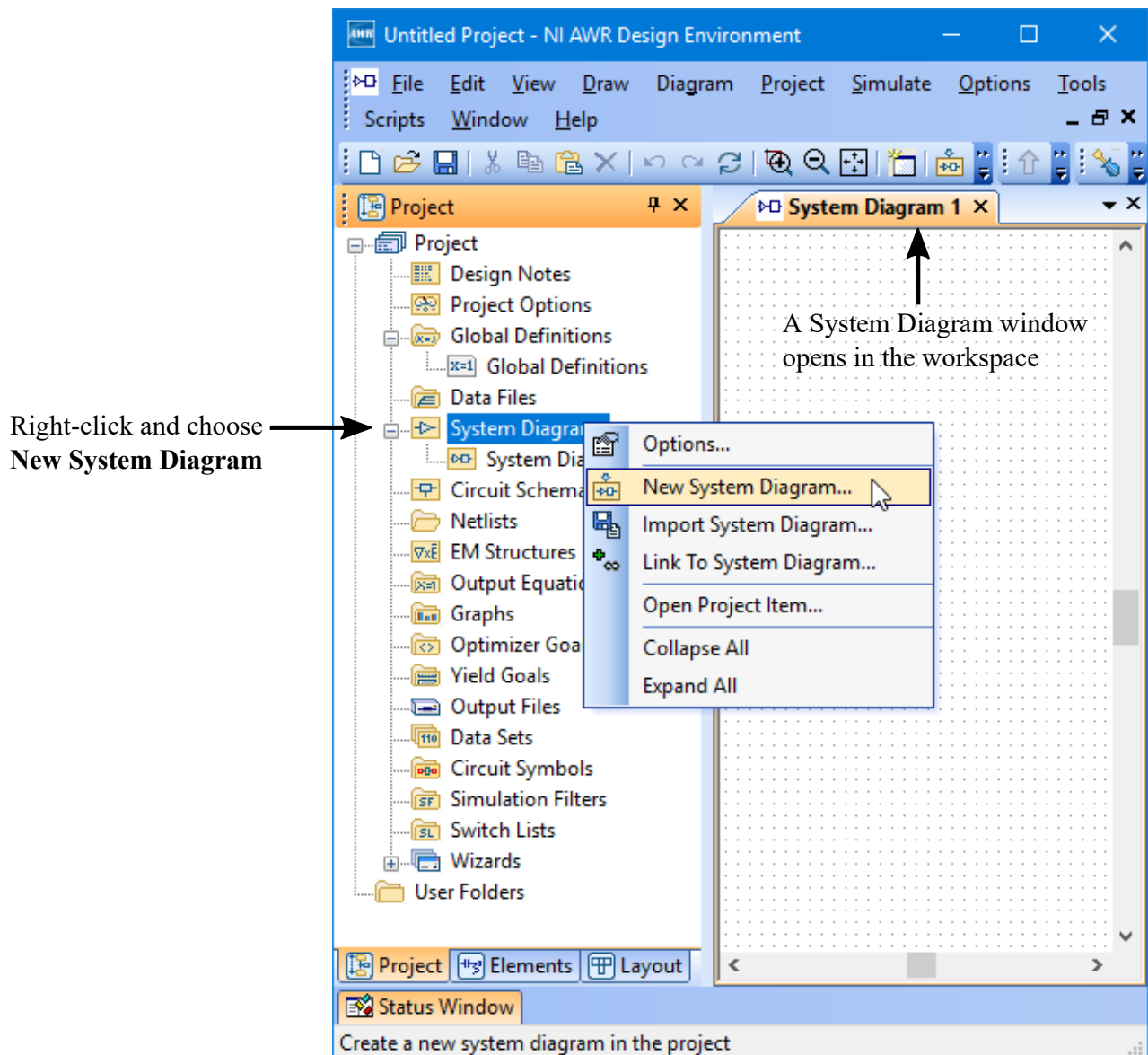
To create a netlist, right-click **Netlists** in the Project Browser, choose **New Netlist**, and then specify a netlist name and type.

After you name the schematic or netlist, a window for it opens in the workspace and the Project Browser displays the new item as a subnode under **Circuit Schematics** or **Netlists**. In addition, the menu bar and toolbar display new command choices and buttons particular to building and simulating schematics or netlists.



Working with System Diagrams in VSS

To create a system diagram, right-click **System Diagrams** in the Project Browser and choose **New System Diagram**, and then specify a system diagram name.



After you name the system diagram, a window for it opens in the workspace and the Project Browser displays the new item as a subnode under **System Diagrams**. In addition, the menu bar and toolbar display new command choices and buttons particular to building and simulating systems.

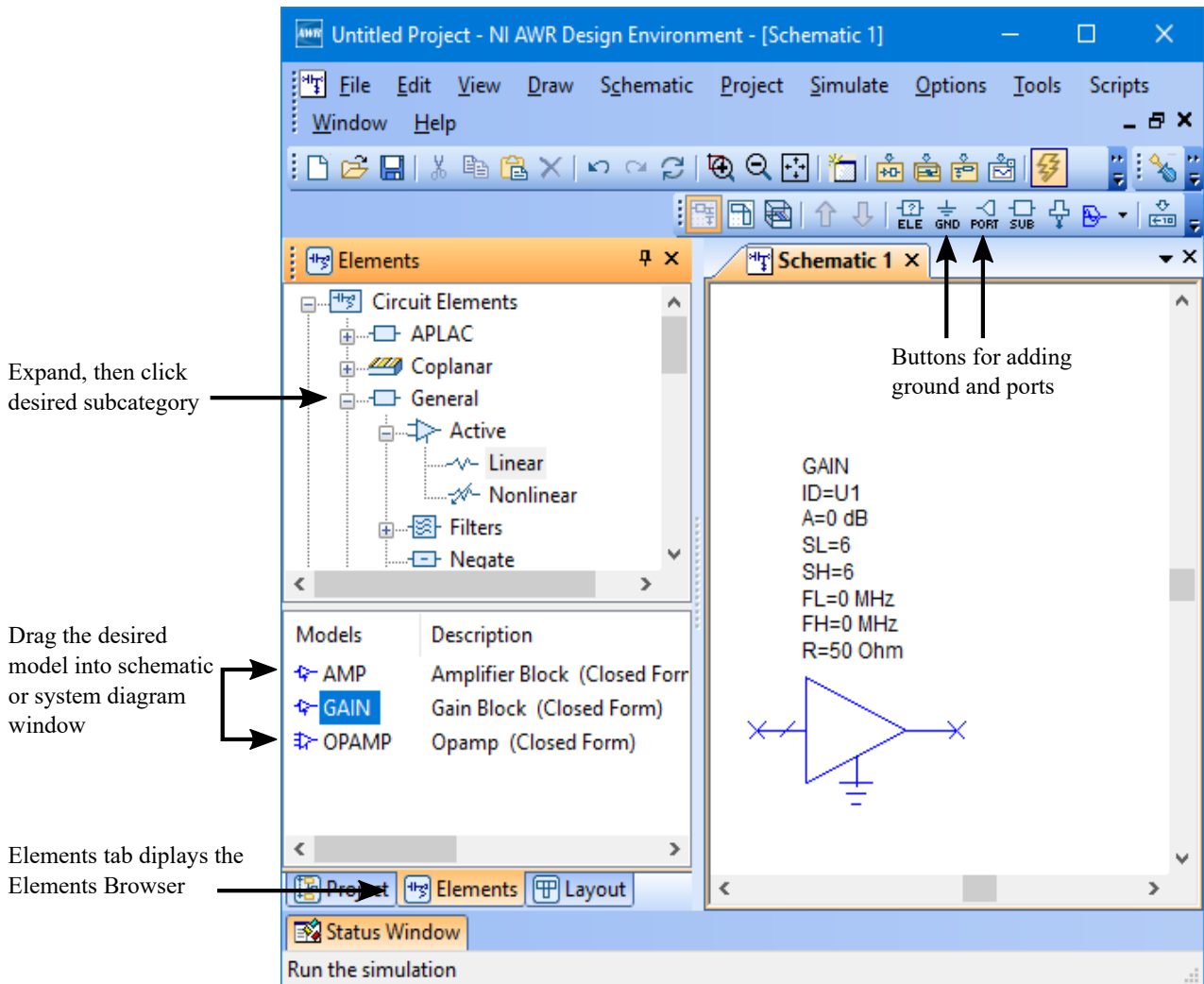
Using the Element Browser

The Element Browser gives you access to a comprehensive database of hierarchical groups of circuit elements for schematics and system blocks for system diagrams. The Libraries folder in the Element Browser provides a wide range of electrical models and S-parameter files from manufacturers.

Circuit elements include models, sources, ports, probes, measurement devices, data libraries, and model libraries that can be placed in a circuit schematic for linear and non-linear simulations.

System blocks include channels, math tools, meters, subcircuits, and other models for system simulations.

- To view elements or system blocks, click the **Elements** tab. The Element Browser replaces the Project Browser window.
- To expand and collapse the model categories, click the **+** or **-** symbol to the left of the category name to view or hide its subcategories. When you click on a category/subcategory, the available models display in the lower window pane. If there are more models than the window can show, a vertical scroll bar displays to allow you to scroll down to see all of the models.
- To place a model into a schematic or system diagram, simply click and drag it into the window, release the mouse button, right-click to rotate it if needed, position it, and click to place it.
- To edit model parameters, double-click the element graphic in the schematic or system diagram window. An Element Options dialog box displays for you to specify new parameter values. You can also edit individual parameter values by double-clicking the value in the schematic or system diagram window and entering a new value in the text box that displays. Press the **Tab** key to move to the next parameter when editing.



NOTE: Choose **Draw > More Elements** to display the Add Circuit Element or Add System Block dialog box to search for elements. Press the **Ctrl** key while clicking a column header to change which column is used to filter.

Adding Subcircuits to Schematics

Subcircuits allow you to construct hierarchical circuits by including a subcircuit block in a schematic (insert a schematic inside of another schematic). The circuit block can be a schematic, a netlist, an EM structure, or a data file.

- To add a subcircuit to a schematic, click **Subcircuits** in the Element Browser. The available subcircuits display in the lower window pane. These include all of the schematics, netlists, and EM structures associated with the project, as well as any imported data files defined for the project.
- To use a data file as a subcircuit, you must first create or add it to the project. To create a new data file, choose **Project > Add Data File > New Data File**. To import an existing data file, choose **Project > Add Data File > Import Data File**. Any new or imported data files automatically display in the list of available subcircuits in the Element Browser.
- To place the desired subcircuit, simply click it and drag it into the schematic window, release the mouse button, position it, and click to place it.
- To edit subcircuit parameters, select the subcircuit in the schematic window, right-click, and choose **Edit Subcircuit**. Either a schematic, netlist, EM structure, or data file opens in the workspace. You can edit it in the same way that you would edit the individual circuit block types.

Adding Subcircuits to System Diagrams

Subcircuits allow you to construct hierarchical systems and to import results of circuit simulation directly into the system block diagram.

- To create a subcircuit to a system diagram, choose **Project > Add System Diagram > New System Diagram** or **Import System Diagram** and then click **Subcircuits** under **System Blocks** in the Element Browser. The available subcircuits display in the lower window pane.
- To place the desired subcircuit, simply click and drag it into the system diagram window, release the mouse button, position it, and click to place it.
- To edit subcircuit parameters, select the subcircuit in the system diagram window, right-click, and choose **Edit Subcircuit**.
- To add a system diagram as a subcircuit to another system diagram, you must first add ports to the system that is designated as a subcircuit.

Adding Ports to Schematics and System Diagrams

To add ports to a schematic or system diagram, expand the **Ports** category in the Element Browser. Under **Circuit Elements** or **System Blocks**, click **Ports** or one of its subgroups, for example, **Harmonic Balance**. The available models display in the lower window pane.

Drag the port into the schematic or system diagram window, right-click to rotate it if needed, position it, and click to place it.

For a shortcut when placing ports and ground, click the **Ground** or **Port** buttons on the toolbar, position the ground or port, and click to place it.

To edit port parameters, double-click the port in the schematic or system diagram window to display an Element Options dialog box.

NOTE: You can change the port type after placing it by double-clicking the port and selecting a **Port type** on the **Port** tab of the dialog box.

Connecting Element and System Block Nodes

You can connect elements directly by positioning the elements so their nodes touch. Small green boxes display to indicate the connection. You can also connect elements with wires.

- To connect element or system block nodes with a wire, position the cursor over a node. The cursor displays as a wire coil symbol. Click at this position to mark the beginning of the wire and slide the mouse to a location where a bend is needed. Click again to mark the bend point. You can make multiple bends.
- Right-click to undo the last wire segment added.
- To start a wire from another wire, select the wire, right-click and choose **Add wire**, then click to mark the beginning of the wire.
- To terminate a wire, click on another element node or on top of another wire.
- To cancel a wire, press the **Esc** key.

Adding Data to Netlists

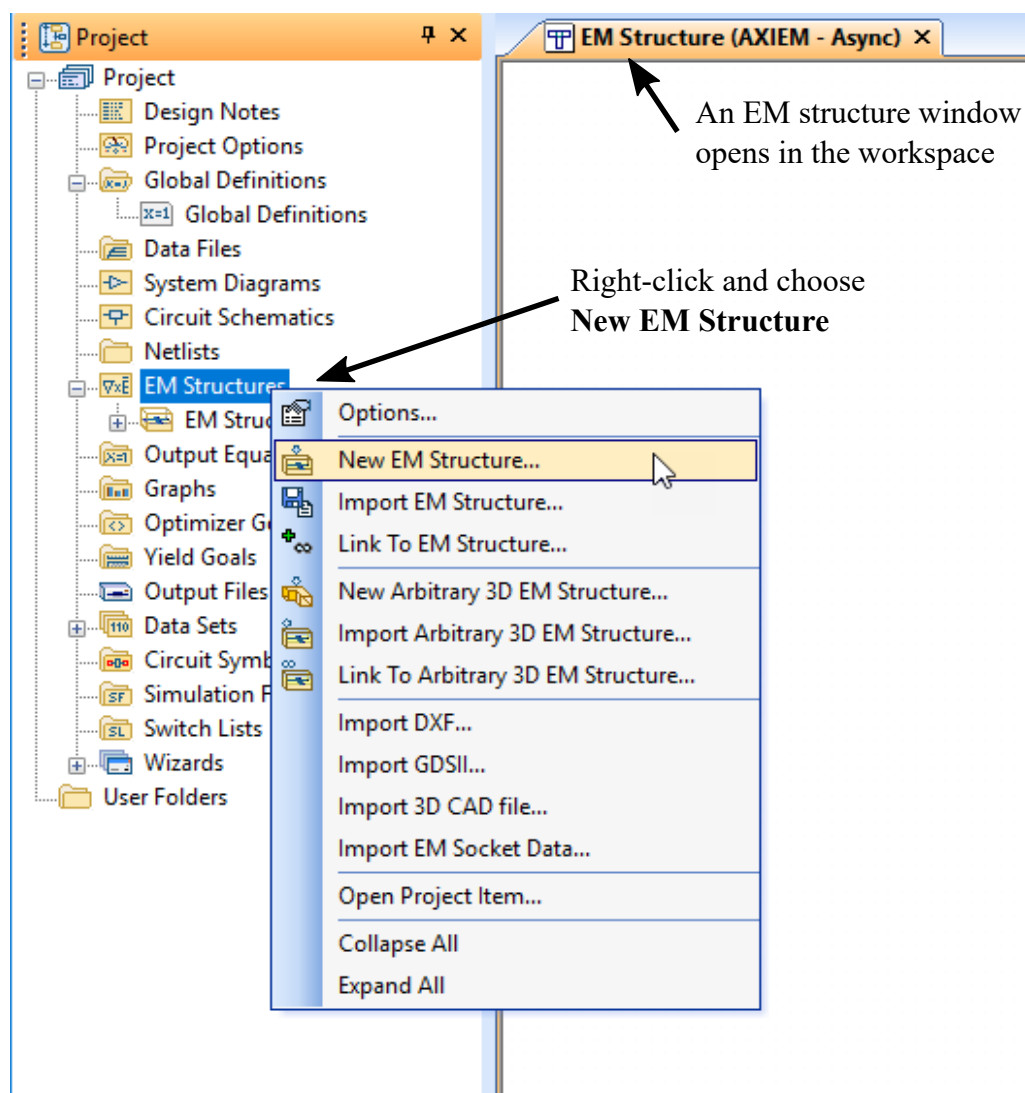
When you create a netlist, an empty netlist window opens into which you type a text-based description of a schematic. Netlist data is arranged in blocks in a particular order, where each block defines a different attribute of an element such as units, equations, or element connections. For more information about creating netlists, see [“Creating a Netlist”](#) in *User Guide*.

Creating EM Structures

EM structures are arbitrary multi-layered electrical structures such as spiral inductors with air bridges.

To create an EM structure, right-click the **EM Structures** node in the Project Browser, and choose **New EM Structure**.

After you specify an EM structure name and select a simulator, an EM structure window opens in the workspace and the Project Browser displays the new EM structure under **EM Structures**. Subnodes of the new EM structure which contain the options that define and describe the EM structure can be displayed as described in [“Working with Schematics and Netlists in Microwave Office”](#). In addition, the menu and toolbar display new choices particular to drawing and simulating EM structures.



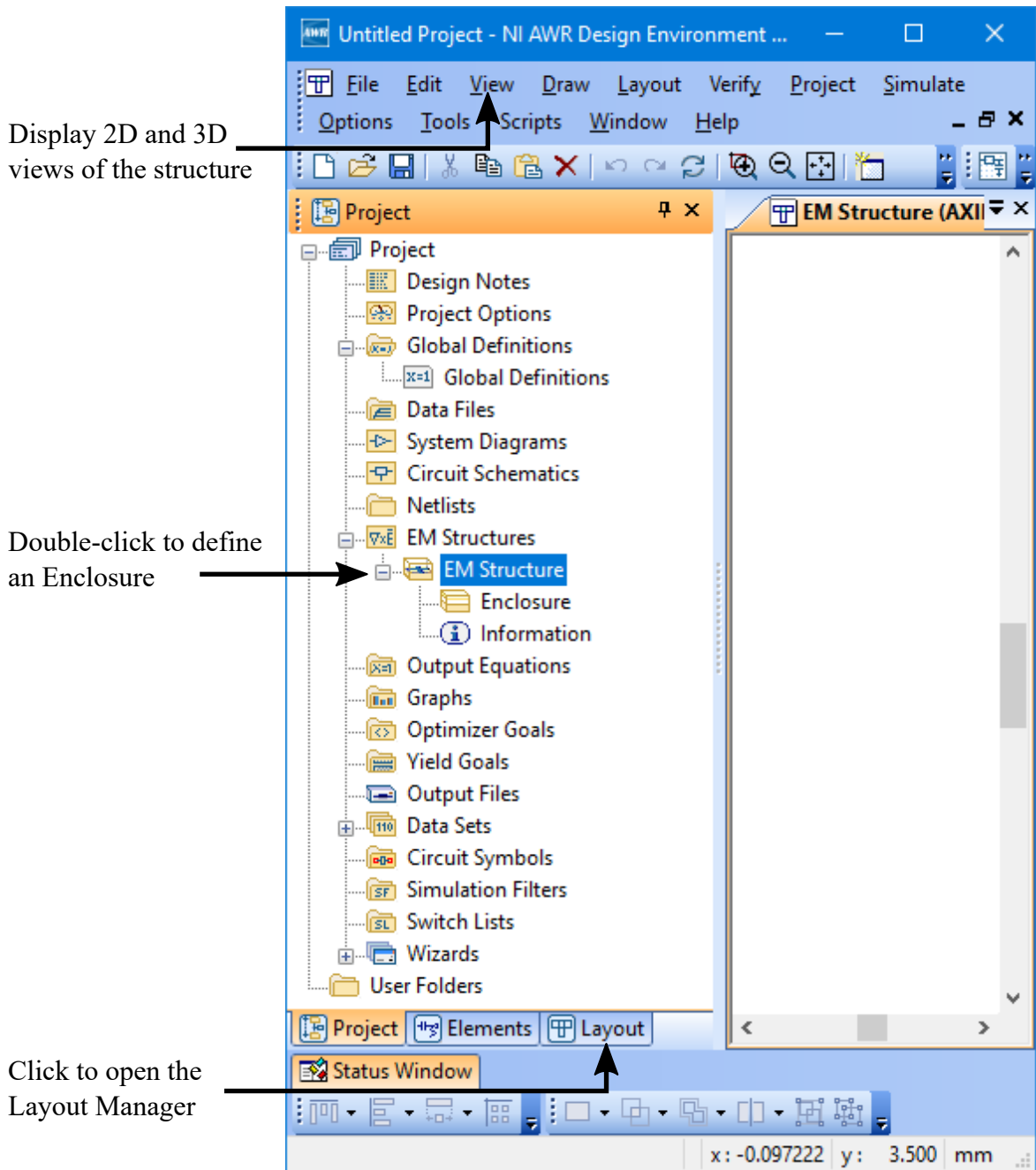
Adding EM Structure Drawings

Before you draw an EM structure, you must define an enclosure. The enclosure specifies things such as boundary conditions and dielectric materials for each layer of the structure.

To define an enclosure, double-click **Enclosure** under your new EM structure in the Project Browser to display a dialog box in which you can specify the required information.

After you define the enclosure, you can draw components such as rectangular conductors, vias, and edge ports in the Layout Manager.

You can view EM structures in 2D (double-click the EM structure node in the Project Browser) and 3D (right-click the EM structure node in the Project Browser and choose **View 3D EM Layout**), and you can view currents and electrical fields using the **Animate** buttons on the EM 3D Layout toolbar.

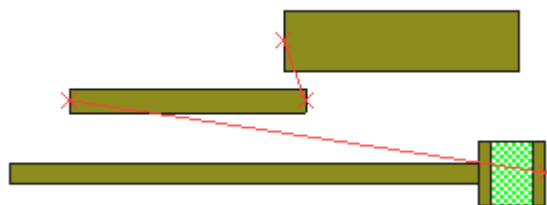


Creating a Layout with Microwave Office

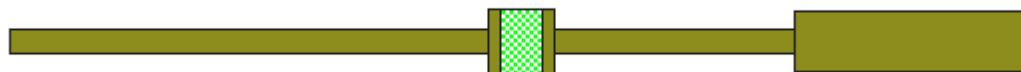
A layout is a view of the physical representation of a circuit, in which each component of the schematic is represented by a layout cell. In the object-oriented NI AWR Design Environment software, layouts are tightly integrated with the schematics and EM structures that they represent, and are simply another view of the same circuits. Any modifications to a schematic or EM structure are automatically and instantly reflected in their corresponding layouts.

To create a layout representation of a schematic, click the schematic window to make it active, then choose **View > Layout**. A layout window tab opens with an automatically-generated layout view of the schematic.

With a schematic window active, you can also click the **View Layout** button on the toolbar to view the layout of a schematic.



The resulting layout contains layout cells representing electrical components floating in the layout window. Choose **Edit > Select All** then choose **Edit > Snap Together** to snap the faces of the layout cells together. The following figure shows the layout view from the previous figure after a snap together operation.

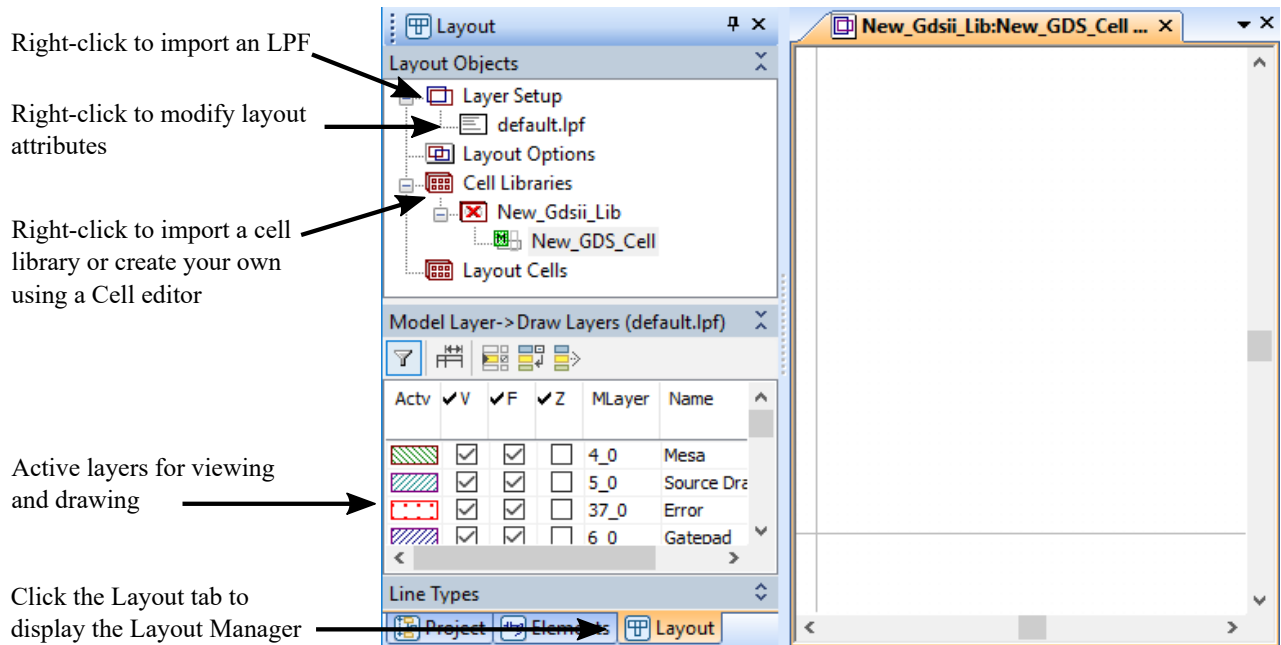


When you choose **View > Layout**, corresponding schematic components with default layout cells are automatically generated for common electrical components such as microstrip, coplanar waveguide, and stripline elements. After the layout is generated, the schematic window displays in blue the components that do not map to default layout cells, and displays in magenta the components that do have default layout cells. You must use the Layout Manager to create or import layout cells for components without them. For more information see [“Using the Layout Manager”](#).

You can draw in the schematic layout window using the Draw tools to build substrate outlines, draw DC pads for biasing, or to add other details to the layout. In this mode, the layout is not part of a schematic element and therefore does not move as part of the snapping process.

Modifying Layout Attributes and Drawing Properties

To modify layout attributes and drawing properties, and to create new layout cells for elements without default cells, click the **Layout** tab to open the Layout Manager.

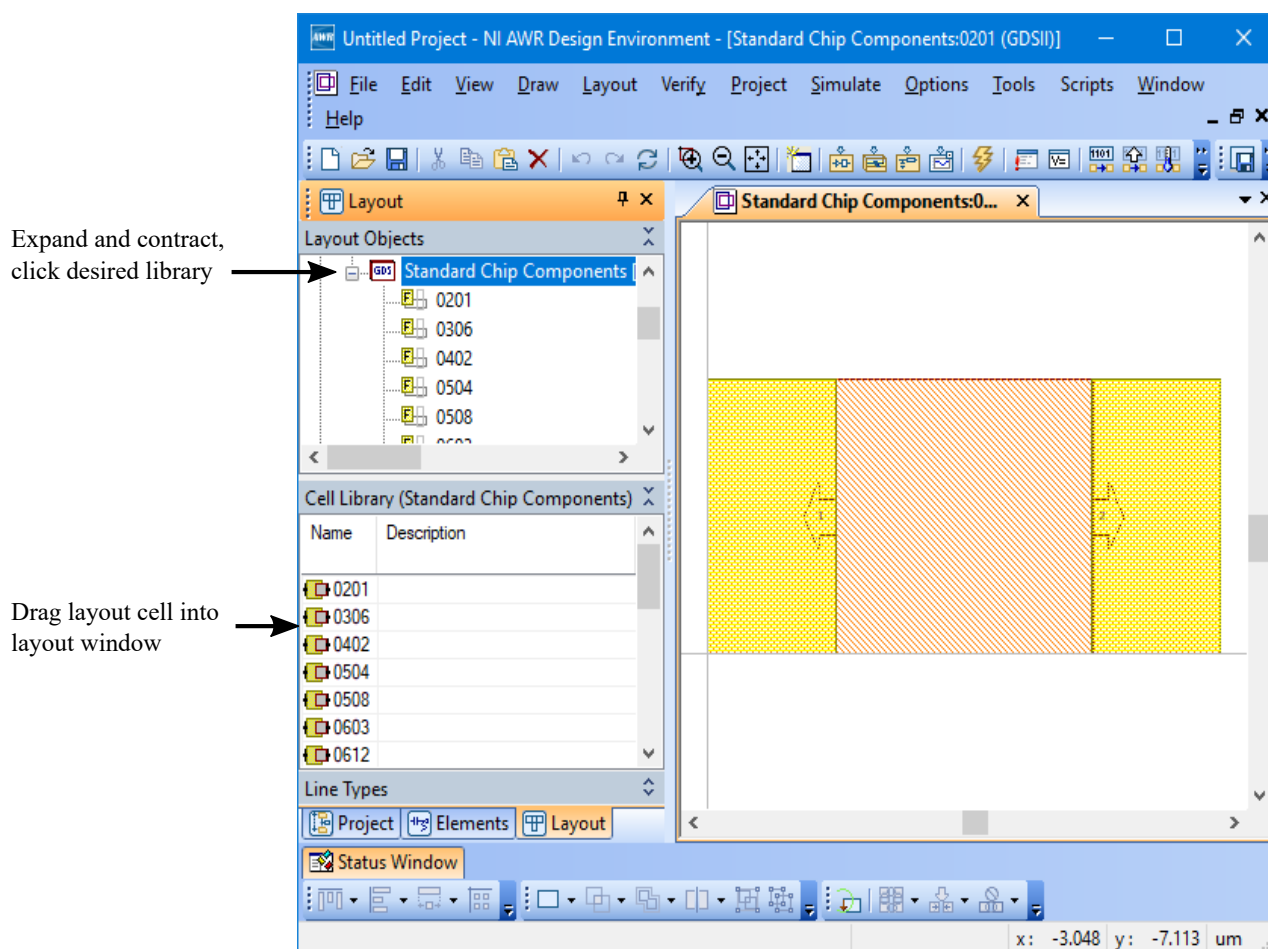


Using the Layout Manager

The **Layer Setup** node in the Layout Manager defines layout attributes such as drawing properties (for example, line color or layer pattern), 3D properties such as thickness, and layer mappings. To modify layer attributes, double-click the node (named "default.lpf" in the previous figure) below the **Layer Setup** node. You can also import a layer process file (LPF) to define these attributes by right-clicking **Layer Setup** and choosing **Import Process Definition**.

The **Cell Libraries** node in the Layout Manager allows you to create artwork cells for elements that do not have default layout cells. The powerful Cell Editor includes such features as Boolean operations for subtracting and uniting shapes, coordinate entry, array copy, arbitrary rotation, grouping, and alignment tools. You can also import artwork cell libraries such as GDSII or DXF into the NI AWR Design Environment suite by right-clicking the **Cell Libraries** node and choosing **Import GDSII Library** or **Import DXF Library**.

After creating or importing cell libraries, you can browse through the libraries and select the desired layout cells to include in your layout. Click the + and - symbols to expand and contract the cell libraries, and click the desired library. The available layout cells display in the lower window pane.



After you define a cell library, you can assign cells to schematic elements. You can also use a cell directly in a schematic layout by clicking and dragging the cell into an open schematic layout window, releasing the mouse button, positioning it, and clicking to place it.

To export a schematic layout to GDSII, DXF, or Gerber formats, click the layout window to make it active, and choose **Layout > Export Layout**. To export a layout cell from the cell libraries, select the cell node in the Layout Manager, right-click and choose **Export Layout Cell**.

Creating Output Graphs and Measurements

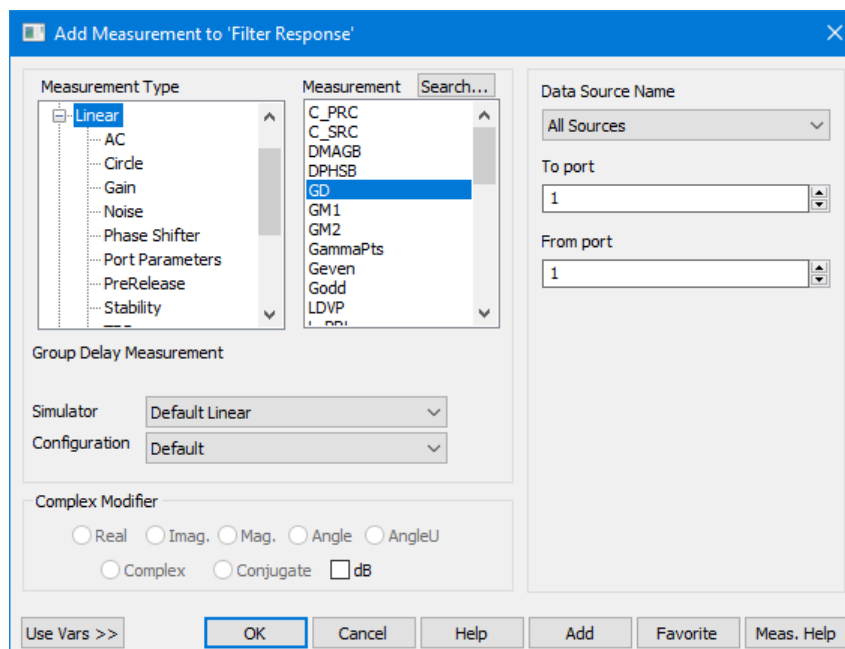
You can view the results of your circuit and system simulations in various graphical forms. Before you perform a simulation, you can create a graph, specifying the data or measurements (for example, gain, noise or scattering coefficients) that you want to plot.

To create a graph, right-click **Graphs** in the Project Browser and choose **New Graph** to display a dialog box in which to specify a graph name and graph type. An empty graph displays in the workspace and the graph name displays under **Graphs** in the Project Browser. The following graph types are available:

Graph Type	Description
Rectangular	Displays the measurement on an x-y axis, usually over frequency.

Graph Type	Description
Constellation	Displays the in-phase (real) versus the quadrature (imaginary) component of a complex signal.
Smith Chart	Displays passive impedance or admittances in a reflection coefficient chart of unit radius.
Polar	Displays the magnitude and angle of the measurement.
Histogram	Displays the measurement as a histogram.
Antenna Plot	Displays the sweep dimension of the measurement as the angle and the data dimension of the measurement as the magnitude.
Tabular	Displays the measurement in columns of numbers, usually against frequency.
3D Plot	Displays the measurement in a 3D graph.

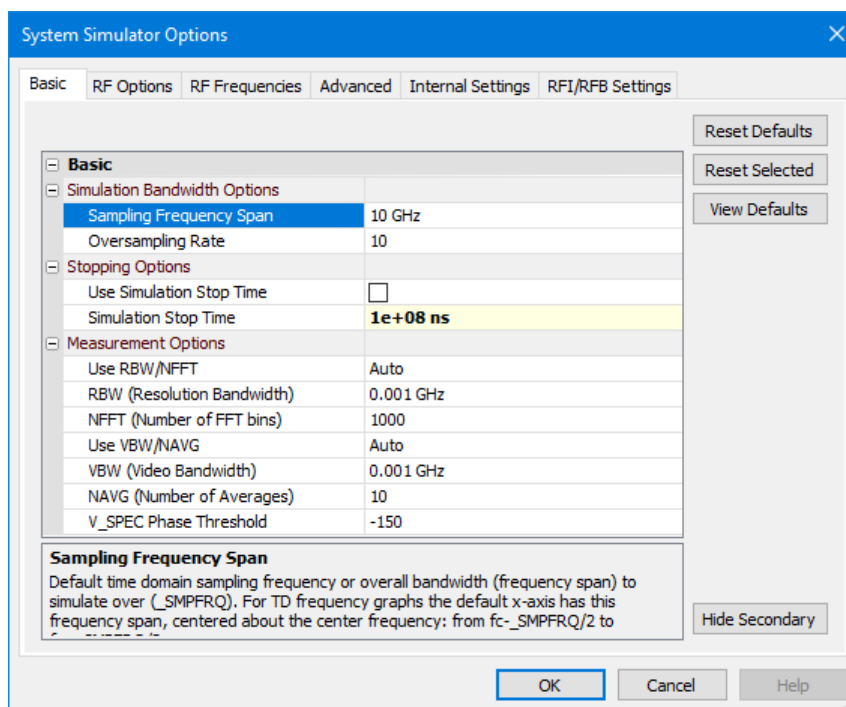
To specify the data that you want to plot, right-click the new graph name in the Project Browser and choose **Add Measurement**. An Add Measurement dialog box similar to the following displays to allow you to choose from a comprehensive list of measurements.



Setting Simulation Frequency and Performing Simulations

To set the Microwave Office simulation frequency, double-click the **Project Options** node in the Project Browser, or choose **Options > Project Options** and then specify frequency values on the **Frequencies** tab in the Project Options dialog box. By default, all the schematics use this frequency for simulation. You can overwrite this frequency with an individual schematic frequency by right-clicking the schematic name under **Circuit Schematics** in the Project Browser and choosing **Options**. Click the **Frequencies** tab, clear the **Use project defaults** check box and then specify frequency values.

To set VSS system simulation frequency, double-click the **System Diagrams** node in the Project Browser or choose **Options > Default System Options**, and then specify frequency values on the **Basic** tab in the System Simulator Options dialog box.



To run a simulation on the active project, choose **Simulate > Analyze**. The simulation runs automatically on the entire project, using the appropriate simulator (for example, linear simulator, harmonic balance nonlinear simulator, or 3D-planar EM simulator) for the different documents of the project.

When the simulation is complete, you can view the measurement output on the graphs and easily tune and/or optimize as needed.

You can perform limited simulations by right-clicking the **Graphs** node or its subnodes to simulate only the graphs that are open, only a specific graph, or simulate for just one measurement on a graph.

Tuning and Optimizing Simulations

The real-time tuner lets you see the effect on the simulation as you tune. The optimizer lets you see circuit parameter values and variables change in real-time as it works to meet the optimization goals that you specified. These features are shown in detail in the linear simulator chapter.

You can also click the **Tune Tool** button on the toolbar. Select the parameters you want to tune and then click the **Tune** button to tune the values. *As you tune or optimize, the schematics and associated layouts are automatically updated!* When you re-run the simulation, only the modified portions of the project are recalculated.

Using Command Shortcuts

The use of keyboard command shortcuts (or hotkeys) can greatly increase efficiency within the NI AWR Design Environment suite. Default menu command shortcuts are available for many common actions such as simulation, optimization, and navigating between the Project Browser, Element Browser and Layout Manager. Default shortcuts display on menus or by choosing **Tools > Hotkeys** to display the Customize dialog box where you can also create custom hotkeys.

Using Scripts and Wizards

Scripts and wizards allow you to automate and extend NI AWR Design Environment functions through customization. These features are implemented via the Microwave Office API, a COM automation-compliant server that can be programmed in any non-proprietary language such as C, Visual Basic™, or Java.

Scripts are Visual Basic programs that you can write to do things such as automate schematic-building tasks within the NI AWR Design Environment software. To access scripts, choose **Tools > Scripting Editor** or any of the options on the **Scripts** menu.

Wizards are Dynamic Link Library (DLL) files that you can author to create add-on tools for the NI AWR Design Environment suite; for example, a filter synthesis tool or load pull tool. Wizards display under the **Wizards** node in the Project Browser.

Using Online Help

Online Help provides information on the windows, menu choices, and dialog boxes in the NI AWR Design Environment suite, as well as for design concepts.

To access online Help, choose **Help** from the main menu bar or press the **F1** key anytime during design creation. The Help topic that displays is context sensitive-- it depends on the active window and/or type of object selected. The following are examples:

- Active window = graph, Help topic = "Working with Graphs" topic.
- Active window = schematic (with nothing selected), Help topic = "Schematics and System Diagrams in the Project Browser".
- Active window = schematic (with an element selected), Help topic = the Help page for that element.
- Active window = schematic (with an equation selected), Help topic = "Equation Syntax".
- Active window = schematic layout (with nothing selected), Help topic = "Layout Editing".

Context sensitive Help is also available by:

- clicking the Help button in most dialog boxes
- right-clicking a model or system block in the Element Browser and choosing **Element Help**, or selecting an element in a schematic or a system block in a system diagram and pressing **F1**, or clicking the **Element Help** button in the Element Options dialog box.
- clicking the **Meas Help** button in the Add/Modify Measurement dialog box
- selecting a keyword (for example; object, object model, or Visual Basic syntax), and pressing **F1** for Help in the NI AWR script development environment.

Chapter 3. Microwave Office: Importing Data Files

This chapter includes an example that demonstrates how to import a Touchstone format data file and use it in simulations.

The basic procedures in this example include:

- Importing an S-parameter file
- Making measurements directly on a data file
- Adding a data file to a schematic
- Assigning a symbol to the data file
- Setting up a circuit for analysis

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Importing an S-parameter Data File

In this example you import a Touchstone data file. You can also import other file formats using this procedure.

Creating a New Project

The example you create in this chapter is available in its complete form as *datafile.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl-click** the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. You can use this example file as a reference.

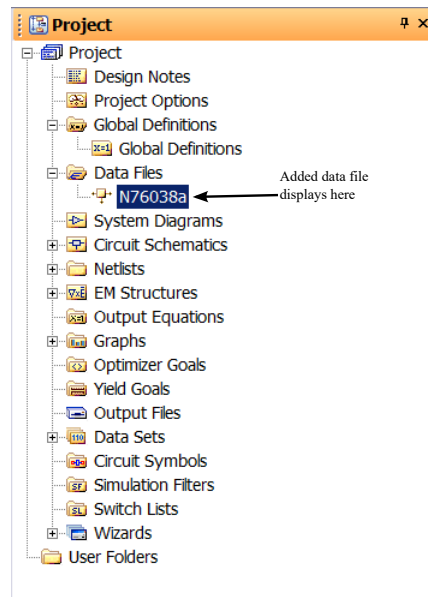
To create a project:

1. Choose **File > New Project**.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**datafile**" as the project name, and then click **Save**.

Importing Data Files

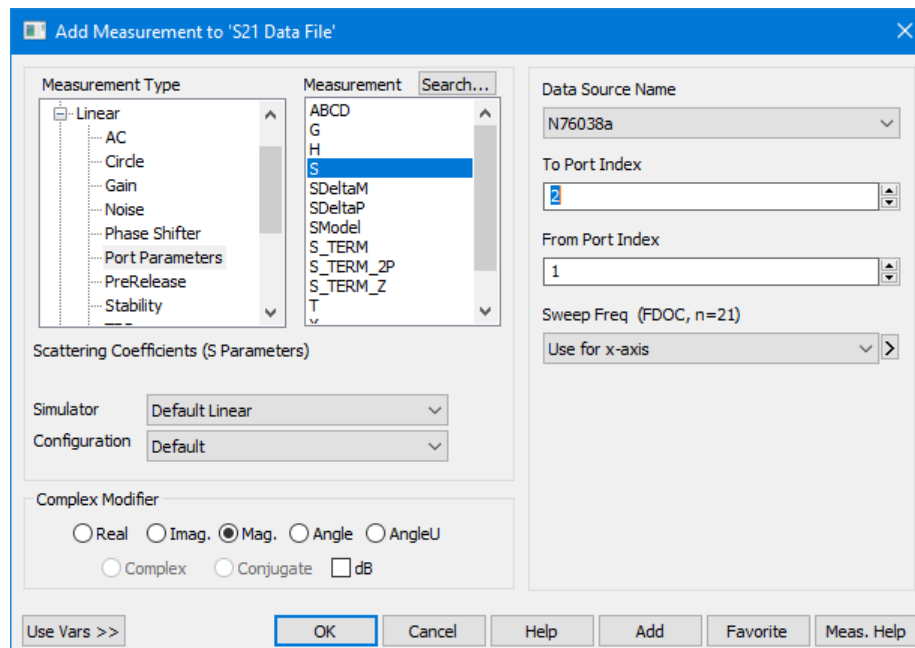
To import a data file:

1. Right-click **Data Files** in the Project Browser and choose **Import Data File**.
2. In the dialog box that displays, browse to the *C:\Program Files\AWR\AWRDE\14\Examples* or *C:\Program Files(x86)\AWR\AWRDE\14\Examples* directory. Select **Touchstone Data Files** as the file type, select the *N73068a.s2p* file, and then click **Open**. The data file is added under the **Data Files** node in the Project Browser as shown in the following figure.

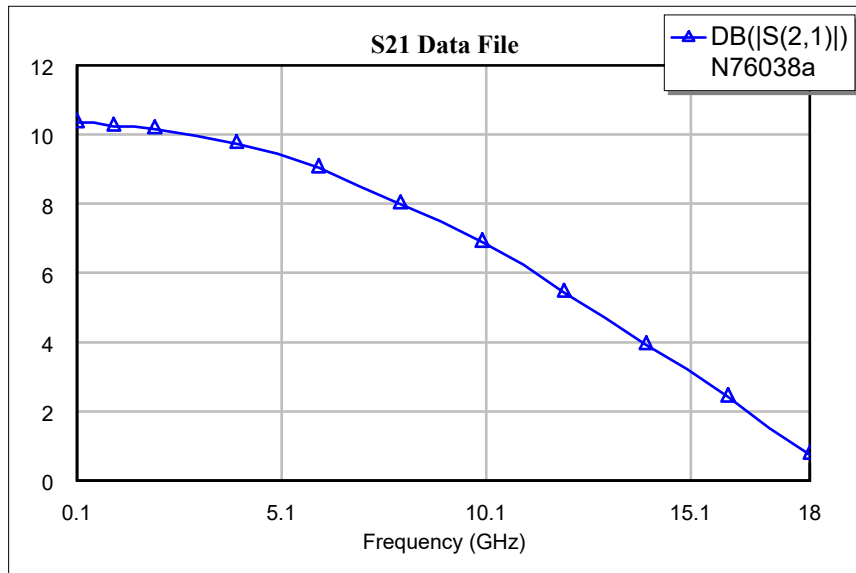


Plotting a Data File Directly

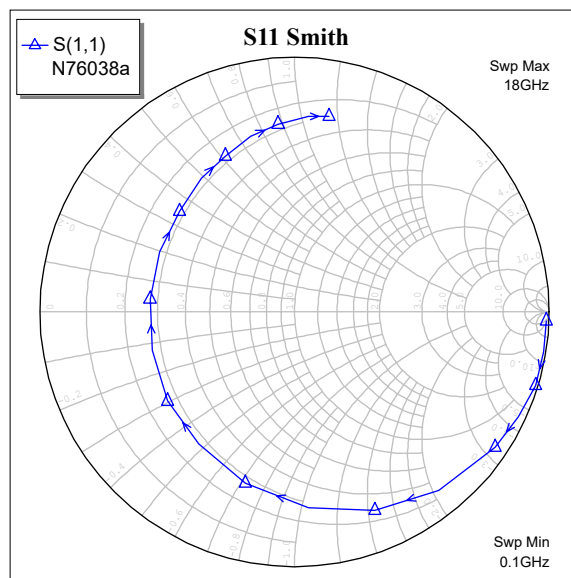
1. Choose **Project > Add Graph** or click the **Add New Graph** button on the toolbar. The New Graph dialog box displays.
2. Type "**S21 Data File**" as the graph name, select **Rectangular** as the graph type, and click **Create**.
3. Right-click the "S21 Data File" graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays. You can also click the **Add New Measurement** button on the toolbar. Add the measurement as shown in the following figure.



- Choose **Simulate > Analyze**. The simulation response as shown in the following figure displays.



- Right-click the "S21 Data File" graph in the Project Browser, and choose **Duplicate as > Smith**. A new Smith Chart named "S21 Data File 1" displays in the Project Browser.
- Right-click "S21 Data File 1" in the Project Browser and choose **Rename**. In the Rename Graph dialog box enter "**S11 Smith**" as the new graph name and then click **OK**.
- Double-click the measurement under the "S11 Smith" chart in the Project Browser to display the Modify Measurement dialog box. Change **To Port Index** to "**1**" and click **OK**.
- Choose **Simulate > Analyze**. The simulation response as shown in the following figure displays.



- Choose **Window > Tile Vertical** to tile the workspace windows.

Adding a Data File to the Schematic

Creating a Schematic

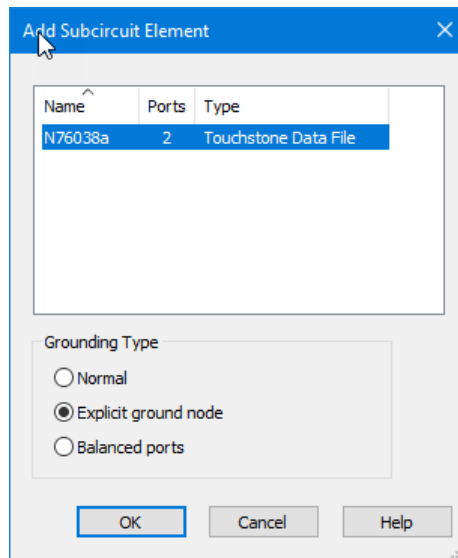
To create a schematic:

1. Choose **Project > Add Schematic > New Schematic**. The New Schematic dialog box displays.
2. Type "Amp" as the schematic name and then click **Create**.

Placing a Data File in a Schematic

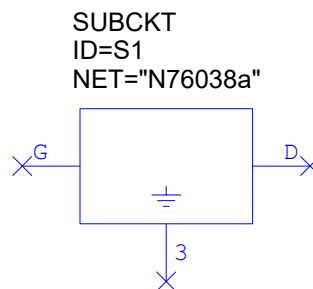
To place a data file in a schematic:

1. Choose **Draw > Add Subcircuit**. The Add Subcircuit Element dialog box displays. Select the data file listed, select **Explicit ground node**, and then click **OK**.

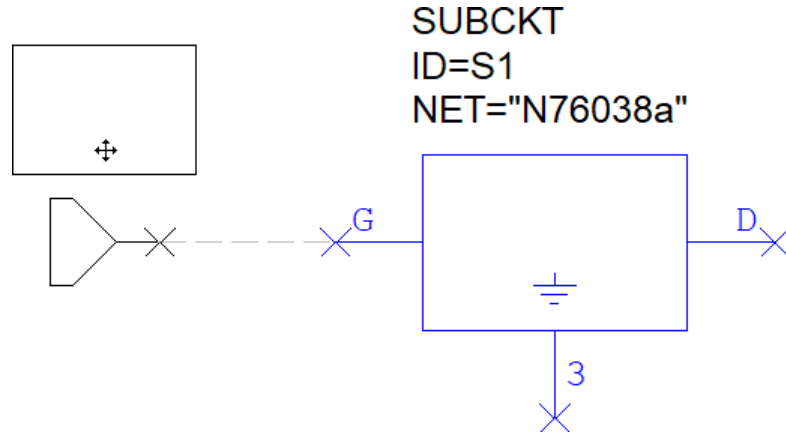


Note that if you don't select **Explicit ground node** the data file shows two pins. You can expose the ground node by right-clicking the subcircuit in the schematic and choosing **Properties > Ground**.

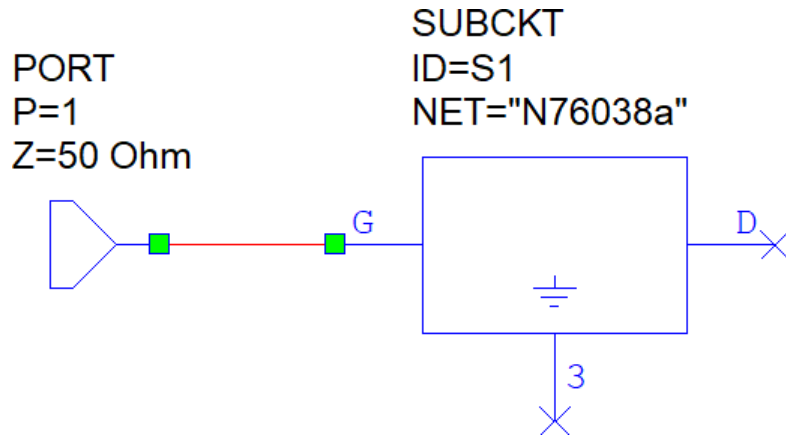
2. Click inside the schematic window to place the data file as a subcircuit. Note the port names "G" and "D" instead of "1" and "2". The names are defined in the data file header. Double-click the "N76038a" data file in the Project Browser to view or edit it.



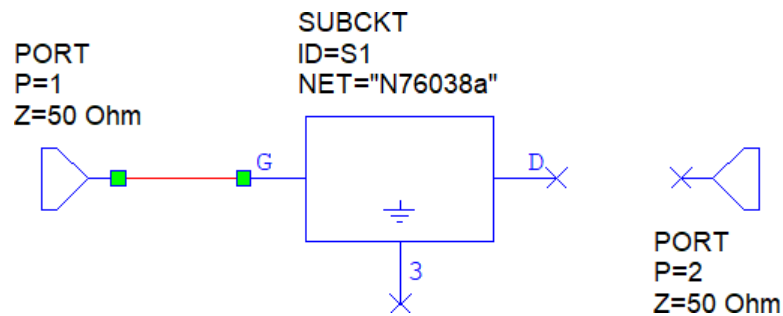
3. Choose **Draw > Add Port** or click the **Port** button on the toolbar.
4. Add a port on the G pin side of the SUBCKT, as shown in the following figure. A dashed wire line displays from the port pin to the G pin when the port aligns to the pin.



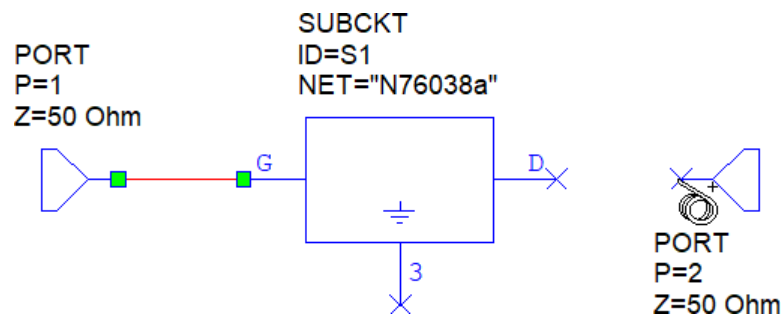
5. **Shift**-click to place the port. The port is automatically wired to the G pin as shown.



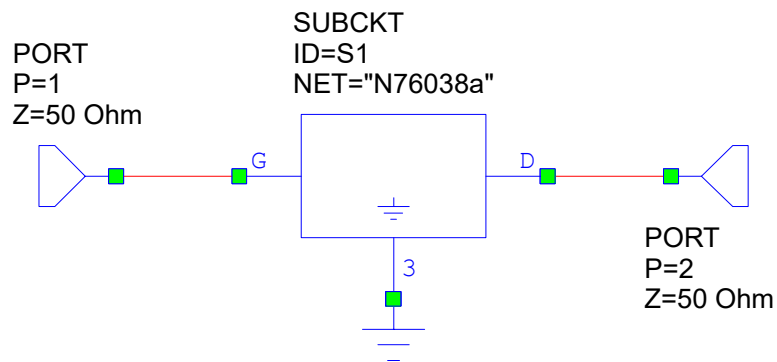
6. Add another port on the other side of the SUBCKT block as shown in the following figure. This time simply click to place the PORT element.



- Place the cursor over the pin of the PORT element. The cursor displays as a wire coil symbol as shown in the following figure.



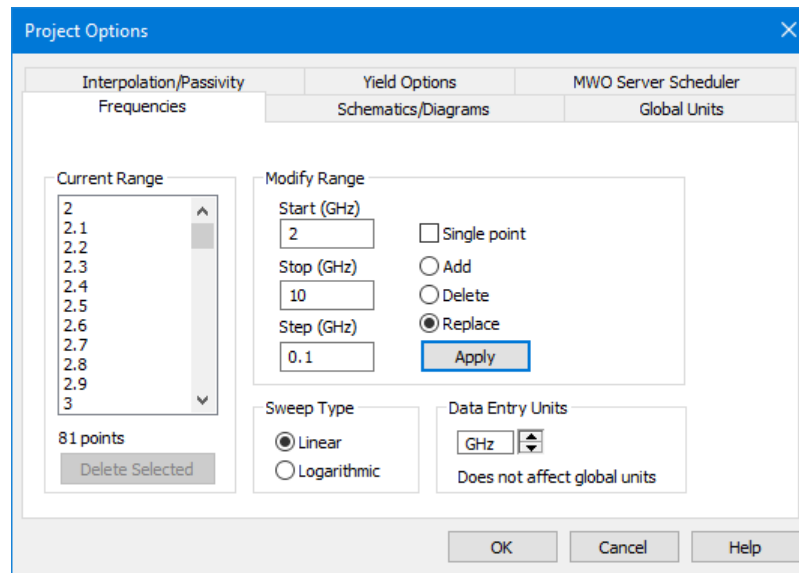
- Click, then drag the wire to pin D of the SUBCKT and click to place it. Complete the wiring of the circuit as shown in the following figure.
- Choose **Draw > Add Ground** or click the **Ground** button on the toolbar. Add a ground as shown in the following figure.



Specifying the Simulation Frequency

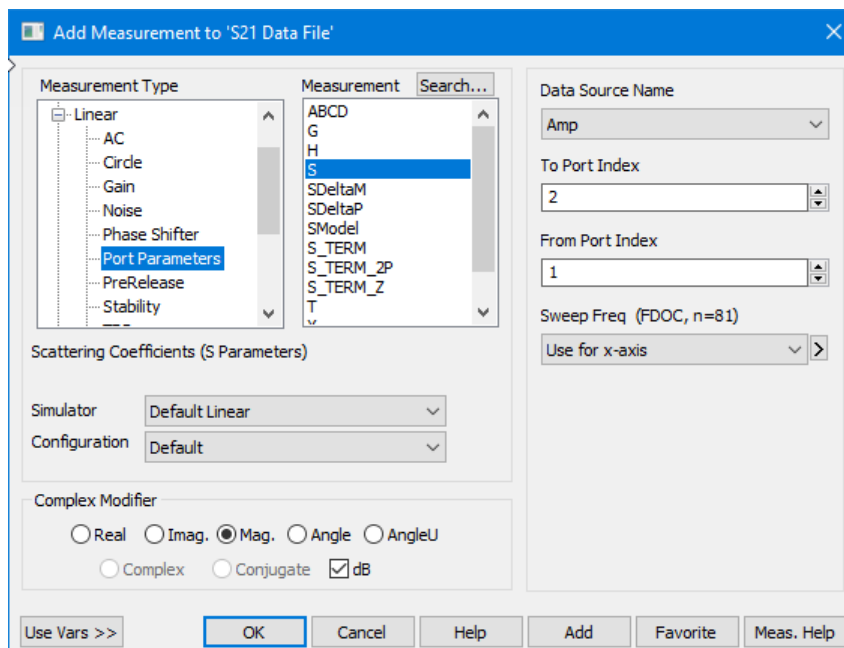
To specify the simulation frequency:

- Choose **Options > Project Options** or double-click the **Project Options** node in the Project Browser.
- Click the **Frequencies** tab.
- Type "2" in **Start**, "10" in **Stop**, and "0.1" in **Step**, and then click **Apply**. The frequency range and steps you specified display in **Current Range**.
- Click **OK**.

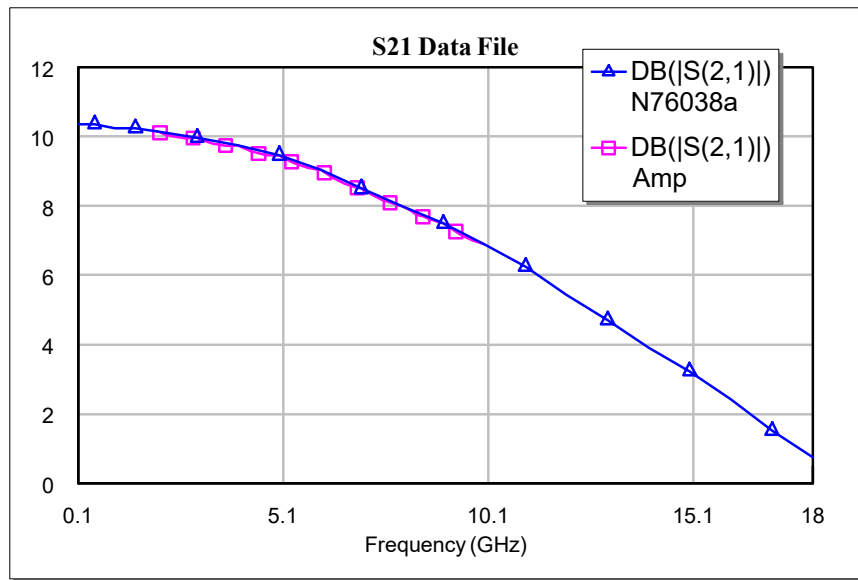


Simulating a Schematic with a Data File

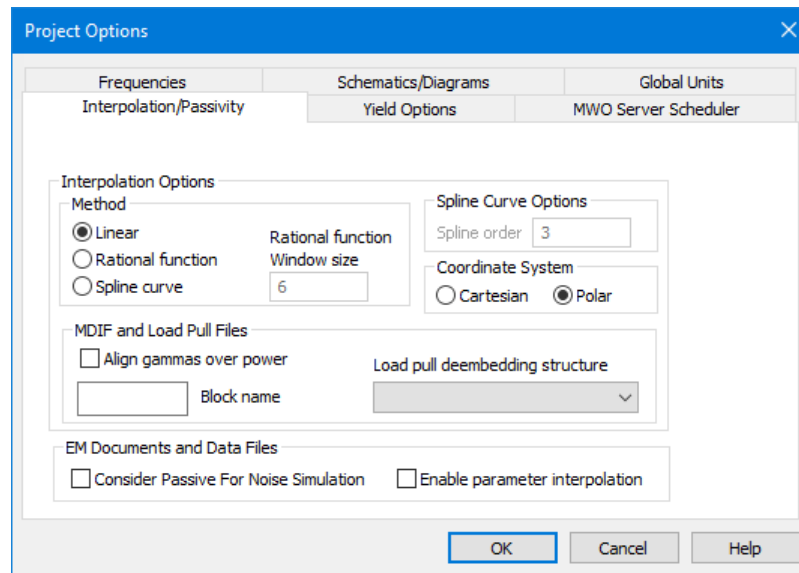
1. Right-click the "S21 Data File" graph in the Project Browser and choose **Add Measurement**, or click the **Add New Measurement** button on the toolbar. The Add Measurement dialog box displays. Add the measurement as shown in the following figure.



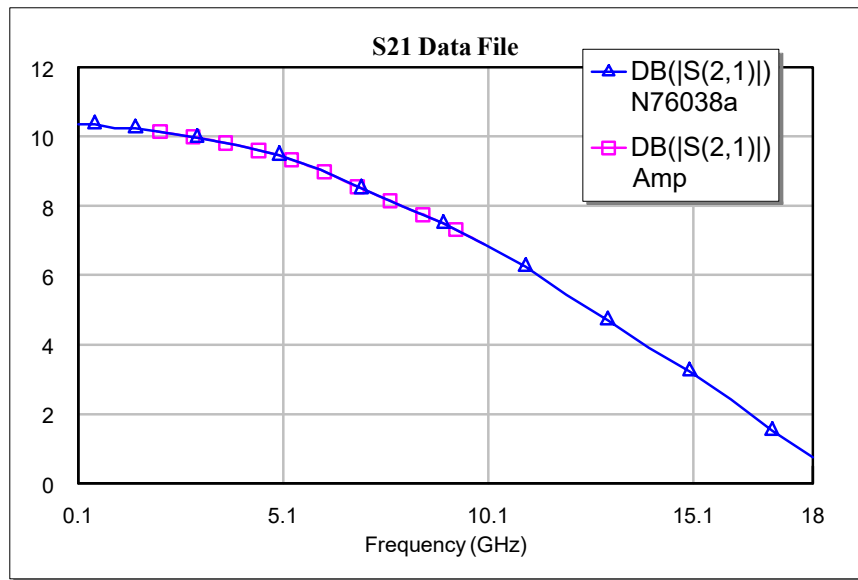
2. Choose **Simulate > Analyze**. The simulation response as shown in the following figure displays. Notice that the simulation result from the schematic looks coarse and doesn't match that of the direct data file measurement.



3. Choose **Options > Project Options**.
4. Click the **Interpolation/Passivity** tab and select **Polar** as the **Coordinate System**, then click **OK**.



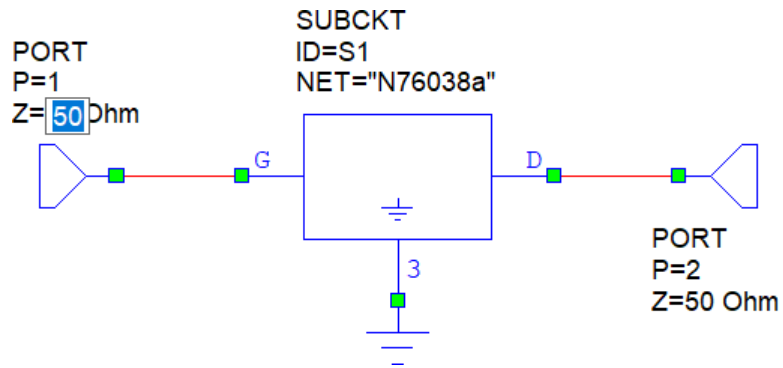
5. Choose **Simulate > Analyze**. The simulation response shown in the following figure displays. Notice the change in trace from the schematic simulation.



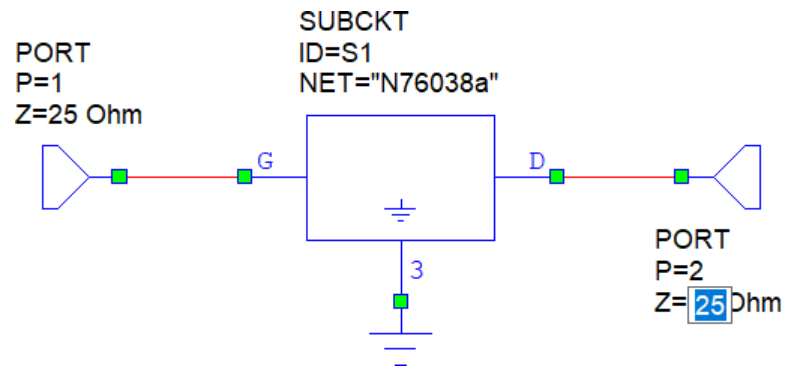
Renormalizing the Data File to a Different Impedance

To renormalize the data to different impedance:

1. In the AMP schematic, double-click the PORT1 Z parameter value to activate edit mode, as shown in the following figure.



2. Type "25" as the Z parameter value for PORT1 and PORT2.



3. Choose **Simulate > Analyze** and view the results in the graph.
4. Save and close the project.

Chapter 4. Microwave Office: Using the Linear Simulator

Linear simulators use nodal analysis to simulate the characteristics of a circuit. Linear simulations are used for circuits such as low noise amplifiers, filters, and couplers whose elements can be characterized by an admittance matrix. Linear simulators typically generate measurements such as gain, stability, noise figure, reflection coefficient, noise circles, and gain circles.

Linear Simulations in Microwave Office

The Microwave Office linear simulator architecture uses object-oriented techniques to enable fast and efficient simulations of linear circuits. One of its trademark features is a real-time tuner, allowing you to see resulting simulations from modifying circuit or component parameters in real-time, in addition to performing optimization and yield analysis.

The following example illustrates some of the key features of the Microwave Office linear simulator.

Creating a Lumped Element Filter

This example demonstrates how to use Microwave Office software to simulate a basic lumped element filter using the linear simulator. It includes the following steps:

- Creating a schematic
- Adding graphs and measurements
- Adding auto-search graph markers
- Analyzing the circuit
- Tuning the circuit
- Creating variables
- Optimizing the circuit

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Creating a New Project

The example you create in this chapter is available in its complete form as *linear_example.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl-click** the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. You can use this example file as a reference.

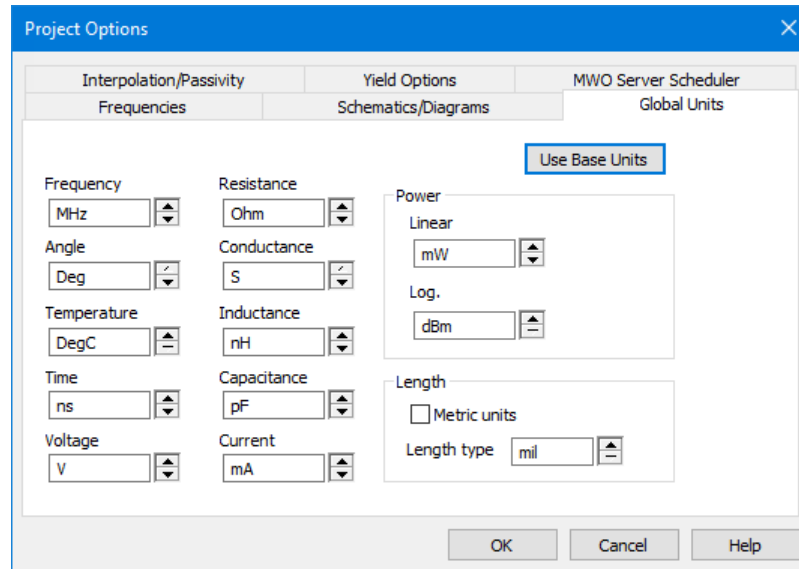
To create a project:

1. Choose **File > New Project**.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**linear_example**" as the project name, and then click **Save**.

Setting Default Project Units

To set default project units:

1. Choose **Options > Project Options**. The Project Options dialog box displays.
2. Click the **Global Units** tab and verify that your settings match those in the following figure. You can choose units by clicking the arrows to the right of the display boxes.



3. Click **OK**. (If a warning message displays, click **OK**).

Creating a Schematic

To create a schematic:

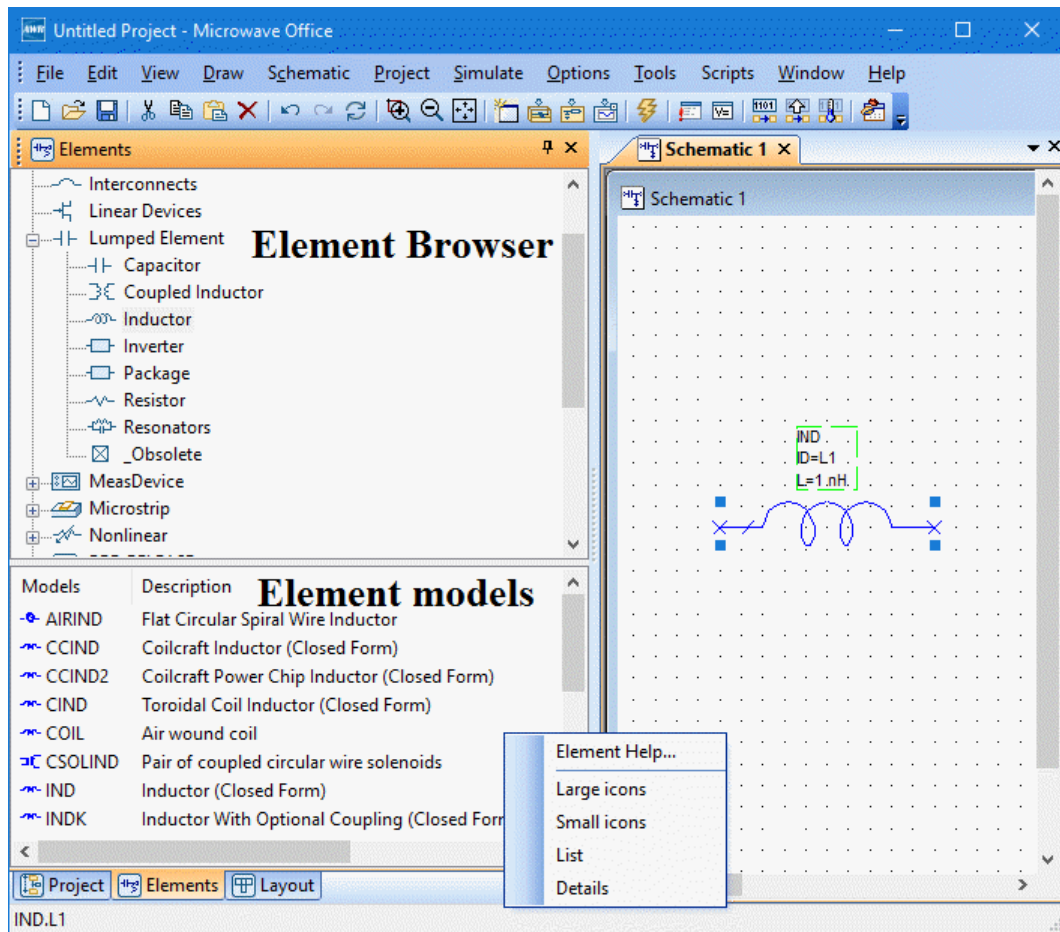
1. Choose **Project > Add Schematic > New Schematic**. The New Schematic dialog box displays.
2. Type "**lpf**", and click **Create**. A schematic window displays in the workspace and the schematic displays under **Circuit Schematics** in the Project Browser.

Placing Elements in a Schematic

Use the scroll arrows along the right and bottom of the schematic window to view different portions of the schematic as you work, or to view the entire schematic choose **View > View All**.

To place elements on a schematic:

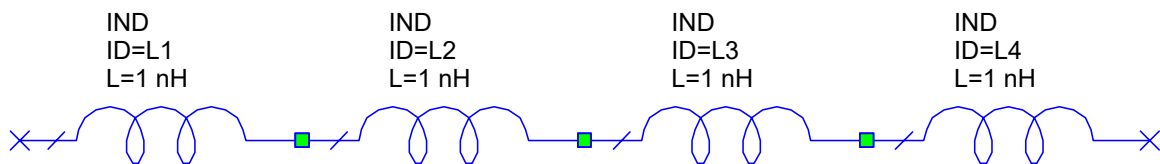
1. Click the **Elements** tab to display the Element Browser. The Element Browser replaces the Project Browser window.
2. If necessary, click the + symbol to the left of the **Circuit Elements** node to expand the elements tree.
3. Under **Circuit Elements**, expand the **Lumped Element** category, then click the **Inductor** group. Select the **IND** model from the bottom window and drag it to the schematic as shown in the following figure.



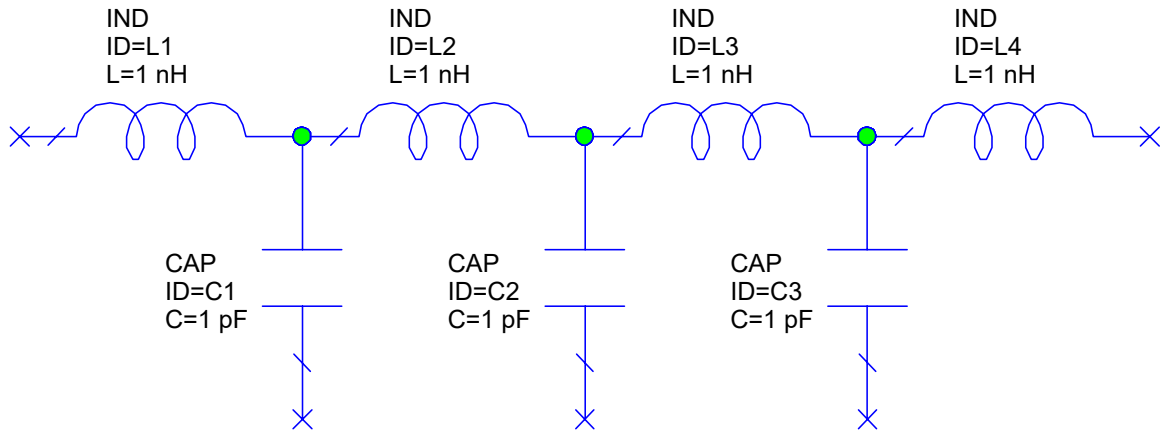
In the Element Browser, you can change the display of the models in the lower window by right-clicking on the window and selecting a model display option. One useful mode is the **Details** option.

4. Add three more IND elements, aligning and connecting each inductor as shown in the following figure.

NOTE: You can also connect elements by moving them to snap their nodes together. When they are properly connected a small colored square (green by default) displays at the connection point and the connection wire extends if you move either element. If you do not see the colored square, try to drag one of the elements into place again.



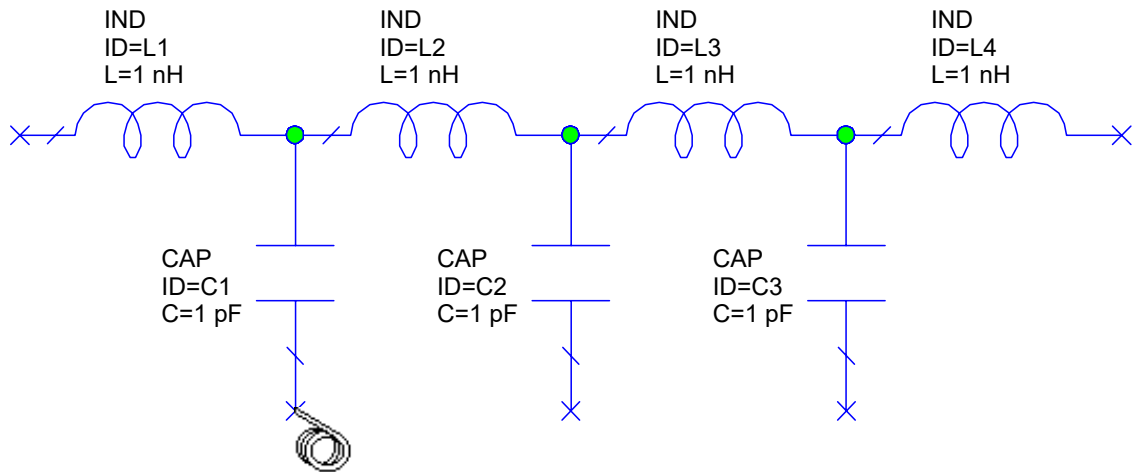
5. Under the **Lumped Element** category, click the **Capacitor** group, then select the CAP model and place it on the schematic as shown in the following figure. Right-click once before placing the capacitor to rotate it as shown.
6. Add two more CAP elements, aligning and linking each capacitor as shown in the following figure.



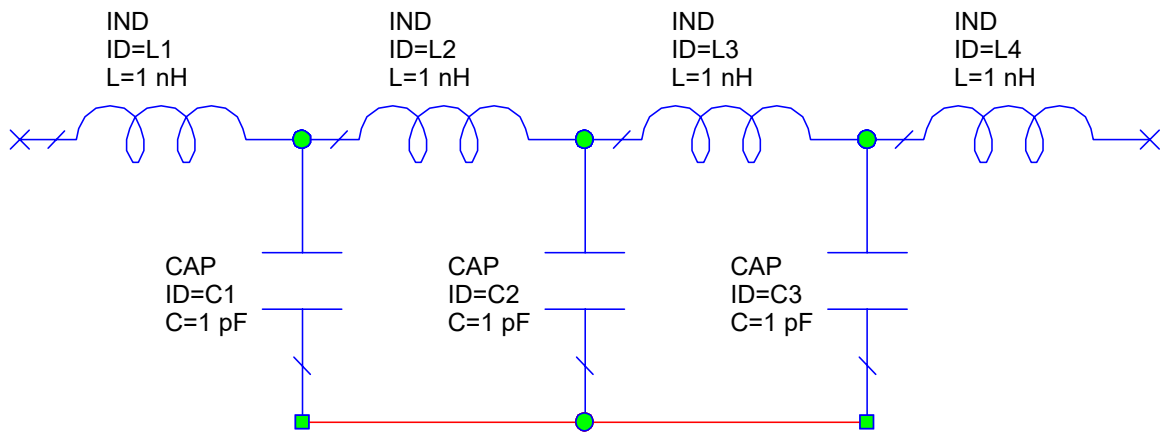
Connecting the Wires

To connect the bottom nodes of the three capacitor elements together:

1. Place the cursor over the bottom node of CAP C1. The cursor displays a wire coil symbol as shown in the following figure.



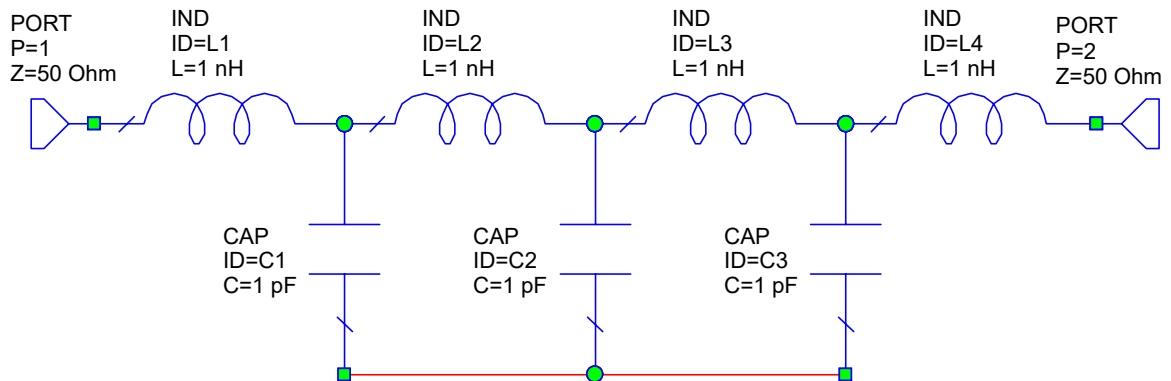
2. Click, then drag the wire past the bottom node of CAP C2, then onto the bottom node of CAP C3, and click to place the wire.



Placing Ports on a Node

To place a port on a node:

1. Choose **Draw > Add Port** or click the **Port** button on the toolbar.
2. Move the cursor onto the schematic, position the port on the first inductor node as shown in the following figure, then click to place it.

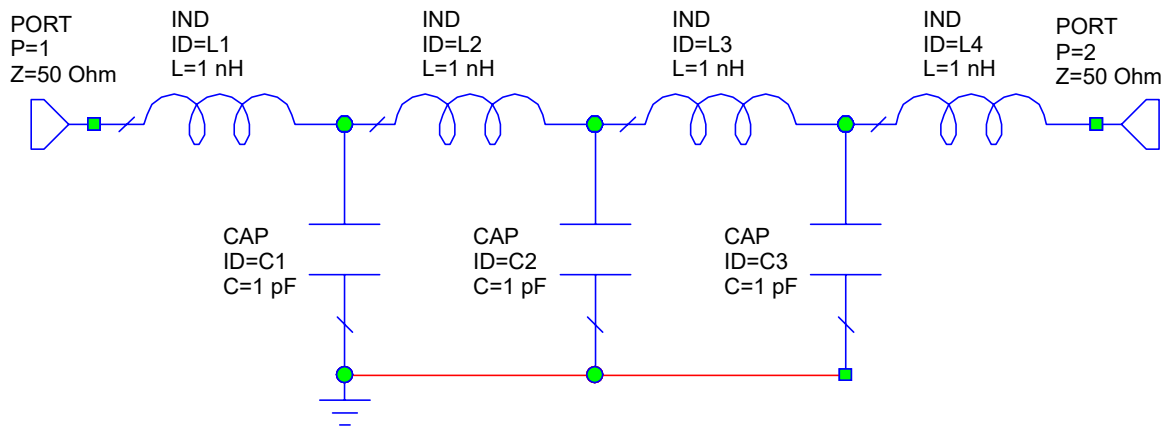


3. Add another port to the right-most inductor, but right-click two times to rotate the port 180-degrees before you place it.

Placing Ground on a Node

To place ground on a node:

1. Choose **Draw > Add Ground**.
2. Move the cursor onto the schematic, position the ground on the bottom node of CAP C1 as shown in the following figure, and click to place it.

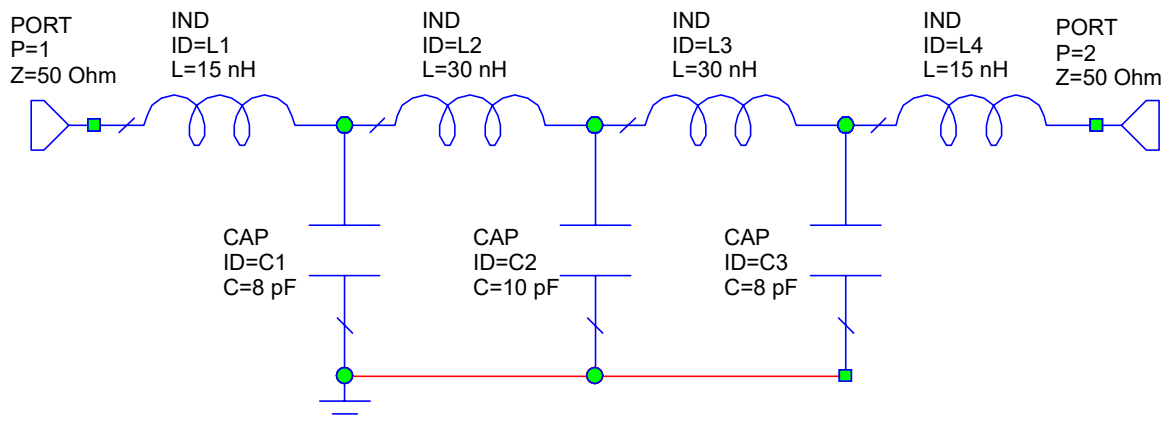


Tip: When adding/moving/pasting an element in a schematic, an inference line displays when nodes between elements align. Press the **Shift** key when placing an inference-aligned element to automatically add the connecting wire.

Editing Element Parameters

To edit the element parameters:

1. Double-click the IND L1 element. The Element Options dialog box displays.
2. Set the L parameter Value to "15" and click **OK**. The change is reflected in the schematic.
3. Repeat Steps 1 and 2 to edit the inductor and capacitor values to match those in the following figure. (To edit capacitor values, set the C parameter value as shown.)



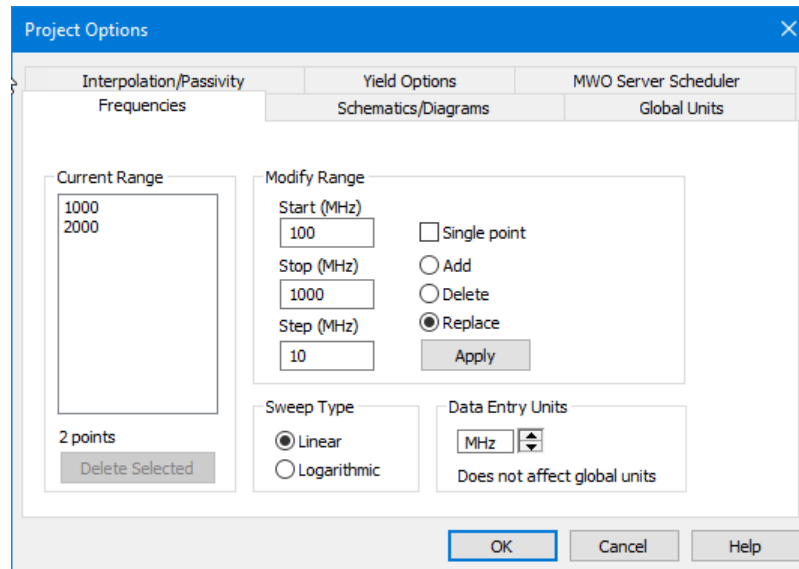
NOTE: Alternatively you can double-click the parameter value directly on the schematic to edit the value in-place.

Specifying the Simulation Frequency

To specify the simulation frequency:

1. Click the **Project** tab.
2. Double-click **Project Options**. The Project Options dialog box displays.

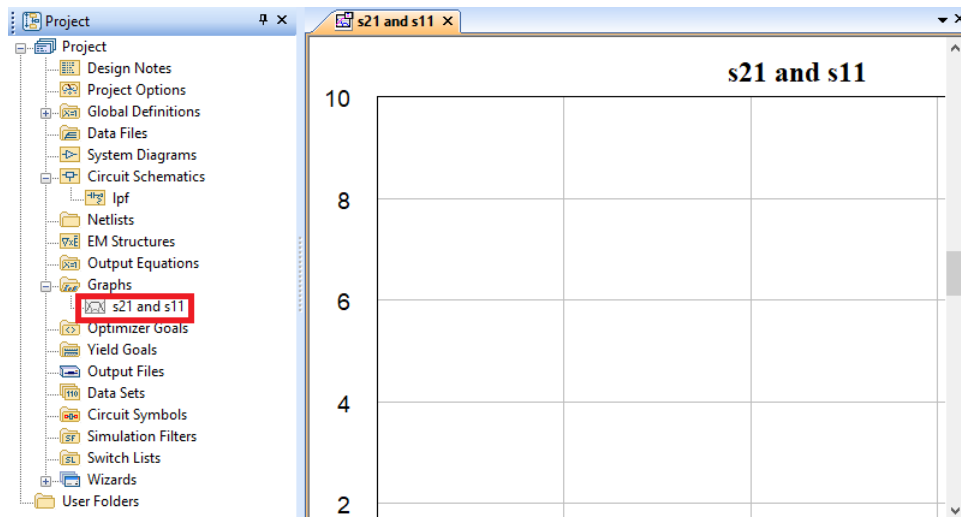
3. Click the **Frequencies** tab.
4. Change the **Data Entry Units** to **MHz**.
5. Type "100" in **Start**, " 1000" in **Stop**, and " 10" in **Step**, and then click **Apply**. The frequency range and steps you specified display in **Current Range**.
6. Click **OK**.



Creating a Graph

To create a graph:

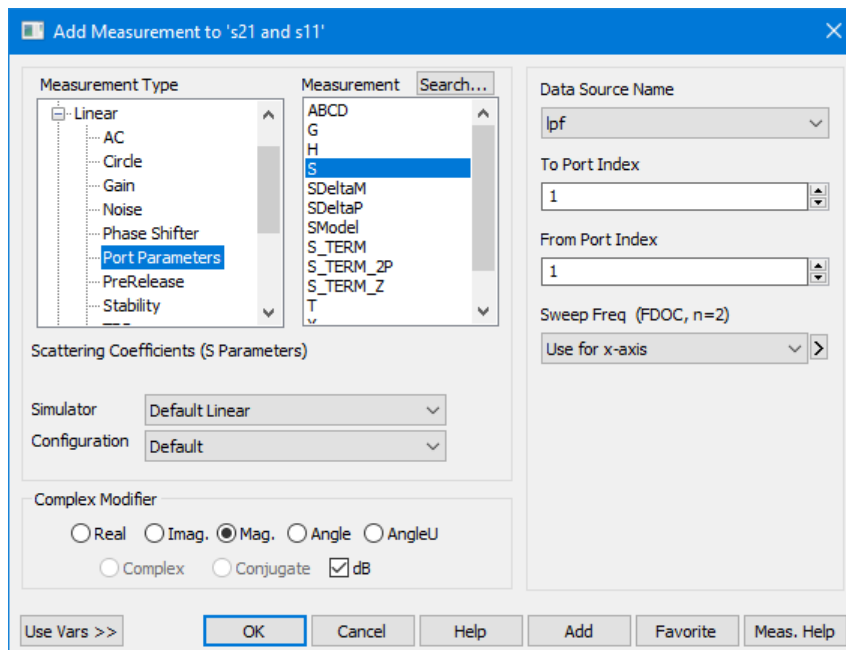
1. Right-click **Graphs** in the Project Browser and choose **New Graph**. You can also click the **Add New Graph** button on the toolbar. The New Graph dialog box displays.
2. Type "s21 and s11" as the graph name, select **Rectangular** as the graph type, and click **Create**. The graph displays in a window in the workspace and displays as a subgroup of **Graphs** in the Project Browser.



Adding a Measurement

To add measurements to the graph:

1. Right-click the "s21 and s11" graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays. You can also click the **Add New Measurement** button on the toolbar.
2. Select **Linear > Port Parameters** as the **Measurement Type** and **S** as the **Measurement**. Click the arrow to the right of **Data Source Name** and select **lpf**. Click the arrows to the right of **To Port Index** and **From Port Index** and select **1** for each. Select **Mag.** as the **Complex Modifier**, select the **dB** check box under **Result Type**, and then click **OK**.

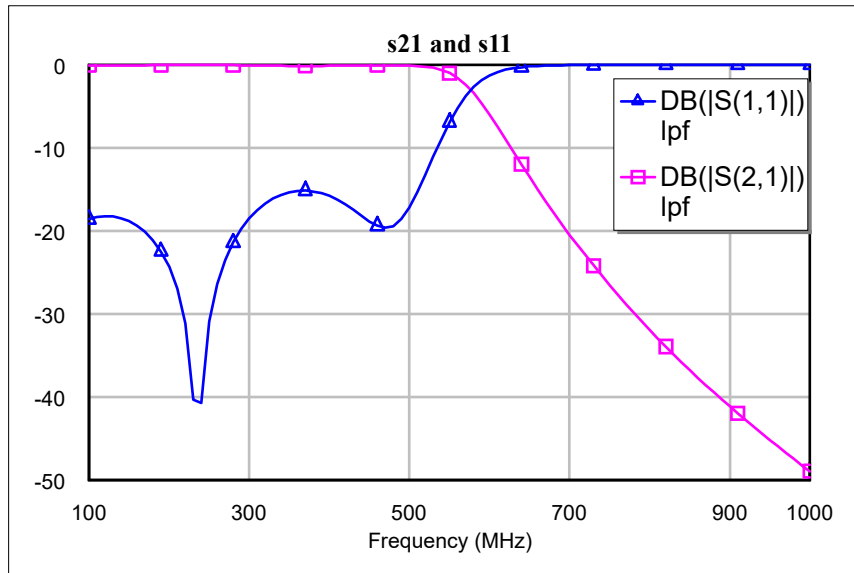


3. Right-click the measurement legend in the graph and choose **Duplicate Measurement**.

4. Change the value in **To Port Index** to "2", and click **OK** to add a second measurement.
5. Click **OK**. The measurements $\text{lpf:DB}(|S(1,1)|)$ and $\text{lpf:DB}(|S(2,1)|)$ display under the "s21 and s11" graph in the Project Browser.

Analyzing the Circuit

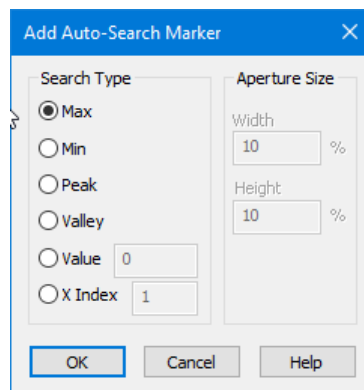
To analyze the circuit choose **Simulate > Analyze**. The simulation response shown in the following figure displays on the graph. You can also click the **Analyze** button on the toolbar to simulate the active project.



Adding Auto-Search Markers

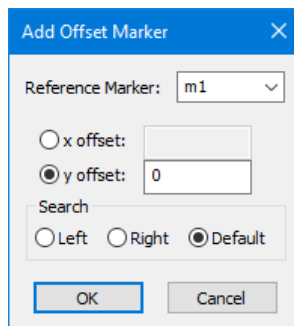
To add auto-search markers:

1. Right-click the graph and choose **Add Auto-Search Marker** to display the Add Auto-Search Marker dialog box.

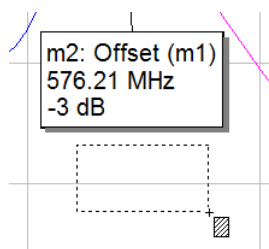


2. Under **Search Type** select **Max**, then click **OK** to close the dialog box.
3. Click anywhere on the $\text{DB}(|S(2,1)|)$ trace to add marker "m1" at the maximum of the trace.

- Right-click the graph and choose **Add Offset Marker** to display the Add Offset Marker dialog box.

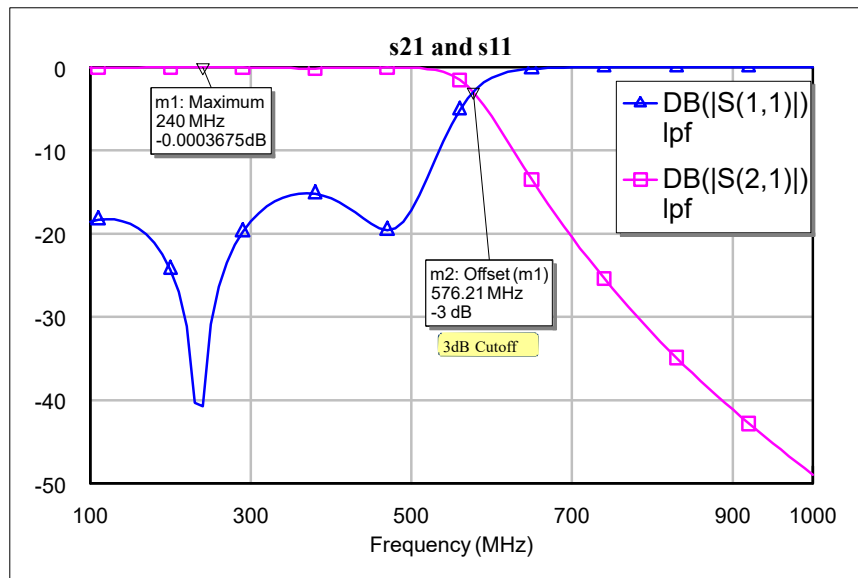


- Set **m1** as the **Reference Marker**, type **"-3"** for the **y offset**, and then click **OK**. Marker **m2** is added to the $DB(|S(2,1)|)$ trace where the y-axis value is 3 dB down from the y-axis value of marker **m1**.
- Right-click on marker "m2" and choose **Add Note**. The cursor changes.
- Click on the graph and drag to form a rectangle, then release the mouse button when the desired note shape is formed.



- Type **"3dB Cutoff"** in the note you created.

The following figure shows the graph with the added auto-search markers. The auto-search markers track with changes to the trace values. Marker "m1" shifts with the trace maximum and marker "m2" shifts with the point on the trace that is 3dB down from marker "m1".

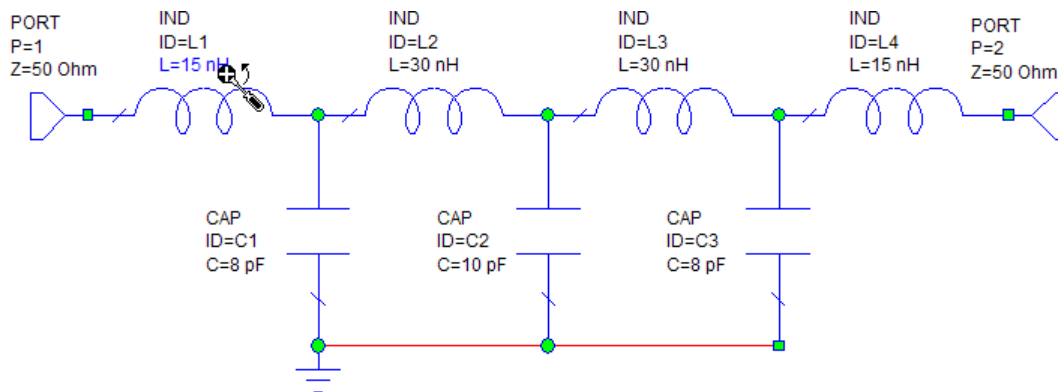


Tuning the Circuit

When you place the Tune tool over a schematic element, the cursor displays as a cross icon to indicate that the parameter is tunable.

To tune the circuit:

1. Right-click the measurement legend and choose **View Source Document** to activate the schematic window.
2. Click the **Tune Tool** button on the toolbar or choose **Simulate > Tune Tool**.
3. Move the cursor over the L parameter of IND L1. The cursor displays as a cross as shown in the following figure.



4. Click to activate the L parameter for tuning. The parameter displays in an alternate color.
5. Click an empty space inside the schematic or click the **Tune Tool** button on the toolbar to deactivate the tuner.
6. Choose **View > Variable Browser** to open the Variable Browser and enable the remaining variables for tuning. Select the **Tune** check box for the IND L4 element and the C parameters of the CAP C1 and CAP C3 elements.
7. Add a tag to the L parameter for inductor elements L1 and L4 by typing "L" in the tag column for each parameter. Also add a "C" tag for the C parameter for capacitors C1 and C3.

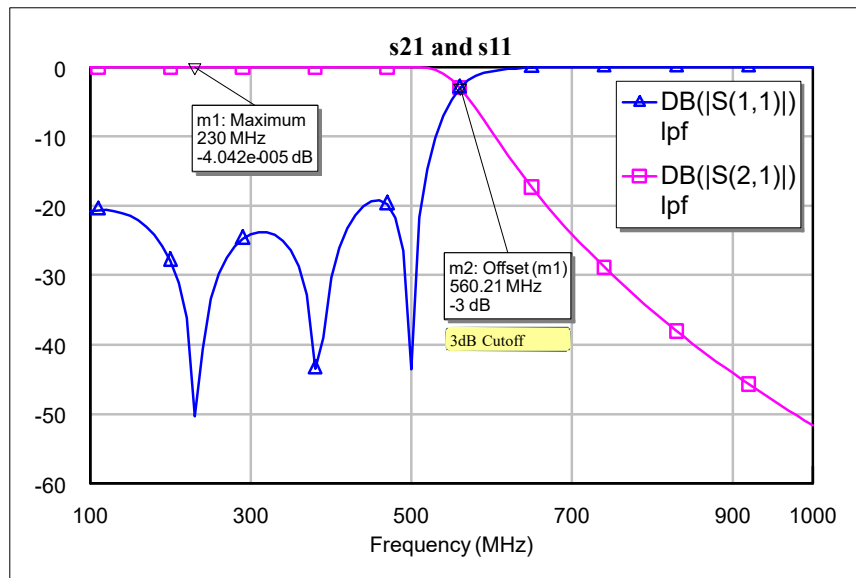
Document	Element	ID	Parameter	Value	✓ Tune	✓ Optimize	✓ Constrain	Lower	Upper	Step Size	Tag
lpf	CAP	C2	C	10	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	CAP	C1	C	8	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				C
lpf	CAP	C3	C	8	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				C
lpf	PORT	P1	Z	50	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	PORT	P2	Z	50	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	IND	L1	L	15	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				L
lpf	IND	L3	L	30	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	IND	L4	L	15	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				L
lpf	IND	L2	L	30	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				

- Click the graph window to make it active.
- Choose **Simulate > Tune**. The Tuner displays. You can also click the **Tune** button on the menu.
- Click a tuning bar and hold down the mouse button while sliding the bar up and down. Note the simulation change on the graph as the variables are tuned. Also note that markers "m1" and "m2" track with simulation changes.
- Click in the **Tag** column on the Tuner and type "L" to filter the choices for tuning down to the inductors. While this is a simple example, it can be useful to tag variables that affect certain performance characteristics of the circuit such as S11, or power added efficiency (PAE).

Document	Element	ID	Parameter	Tune	Step Size	Lower	Tuner	Upper	Value	Tag
lpf	IND	L1	L	<input checked="" type="checkbox"/>	0.2	7.5		30	15	L
lpf	IND	L4	L	<input checked="" type="checkbox"/>	0.2	7.5		30	15	L

- Delete the "L" in the Tuner **Tag** column to see all of the variables enabled for tuning.
- Slide the tuners to the values shown in the following figure, and observe the resulting response on the graph of the tuned circuit. Note that the order of elements in the Tuner may differ from that shown. Click **Simulate > Analyze**.

Document	Element	ID	Parameter	Tune	Step Size	Lower	Tuner	Upper	Value	Tag
lpf	IND	L1	L	<input checked="" type="checkbox"/>	0.2	7.5		30	15	L
lpf	CAP	C1	C	<input checked="" type="checkbox"/>	0.1	4		16	9	C
lpf	CAP	C3	C	<input checked="" type="checkbox"/>	0.1	4		16	9	C
lpf	IND	L4	L	<input checked="" type="checkbox"/>	0.2	7.5		30	15	L



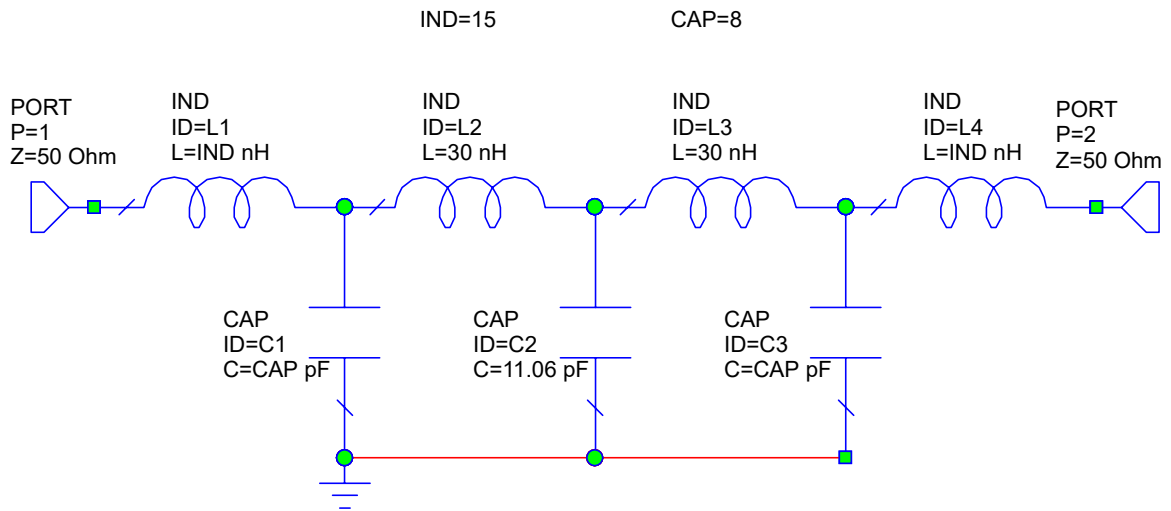
14. Click the **X** at the top right of the Tuner to close it.

Creating Variables

Filters are typically symmetric circuits. To optimize the circuit while maintaining symmetry, you must change some of the parameter values to variables.

To create variables:

1. Click the schematic window to make it active.
2. Choose **Draw > Add Equation** or click the **Equation** button on the toolbar.
3. Move the cursor into the schematic to display an edit box.
4. Position the edit box near the top of the schematic window and click to place it.
5. Type "**IND=15**" (without the quotes) in the edit box, and then click outside of the box.
6. Repeat Steps 2 through 5 to create a second edit box, but type "**CAP=8**" (without the quotes).
7. Double-click the L parameter value of IND L1. An edit box displays. Type the value "**IND**" and then click outside of the edit box.
8. Repeat Step 7 to change the L parameter value of IND L4 to "**IND**", and the C parameter values of CAP C1 and CAP C3 to "**CAP**", as shown in the following figure.



To enable the IND and CAP variables for optimization:

1. Choose **View > Variable Browser**. The Variable Browser dialog box displays.
2. Locate the IND and CAP variables in the **Parameter** column.
3. In the **Optimize** column, click the boxes in the IND, CAP, and CAP C2 rows.

To add constraints to the variables:

1. In the **Constrained** column, click the boxes in the IND, CAP, and CAP C2 rows.
2. In the **Value** column for each of these variables, type "25%" to set the upper and lower constraints to the variable's current value plus or minus twenty-five percent, then click the **X** at the top right of the dialog box to close it. Note that the order of elements displayed may differ from that in the following figure.

Document	Element	ID	Parameter	Value	✓ Tune	✓ Optimize	✓ Constrain	Lower	Upper	Step Size	Tag
lpf	CAP	C2	C	10	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	7.5	12.5		
lpf	IND	L3	L	30	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	PORT	P1	Z	50	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	PORT	P2	Z	50	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				
lpf	EQN		IND	15	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	11.25	18.75		
lpf	EQN		CAP	8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	6	10		
lpf	IND	L2	L	30	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>				

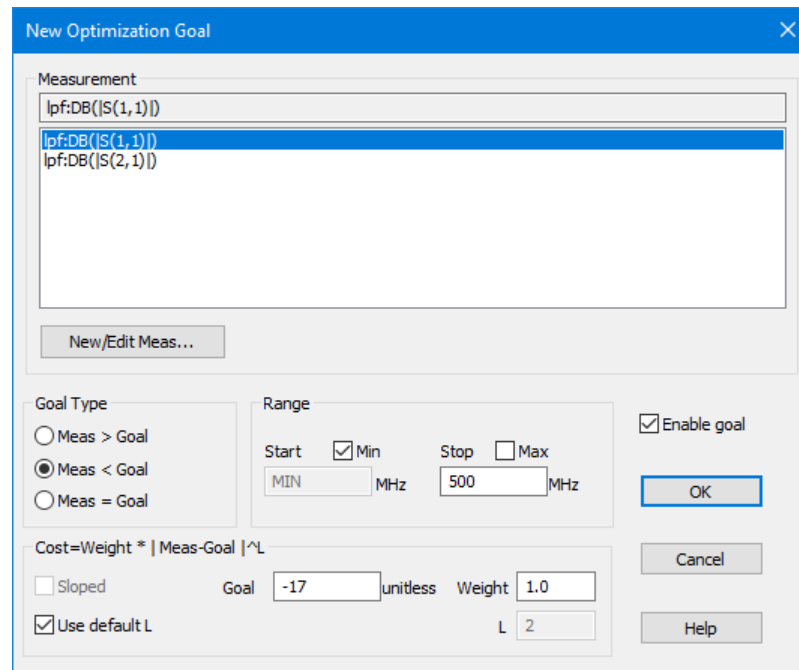
Adding Optimization Goals

When you create an optimization goal, its value is specified in the units of the parameter or measurement used to create the goal.

To add optimization goals:

1. In the Project Browser, right-click **Optimizer Goals** and choose **Add Optimizer Goal**. The New Optimization Goal dialog box displays.

2. Select **lpf:DB(|S(1,1)|)** as the **Measurement**. Select **Meas < Goal** as the **Goal Type**, deselect **Max** under **Range** and type "500" as the **Stop** value, type "-17" as return loss **Goal** in dB, and then click **OK**.

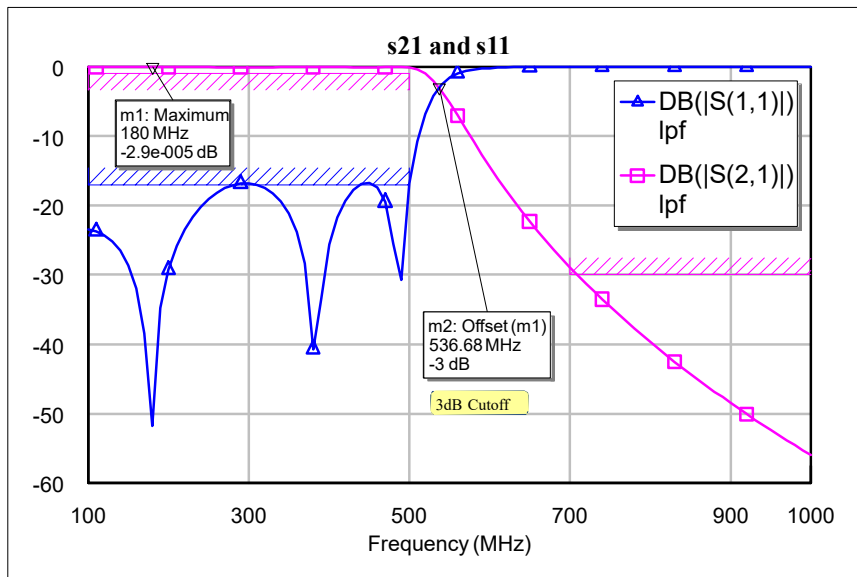
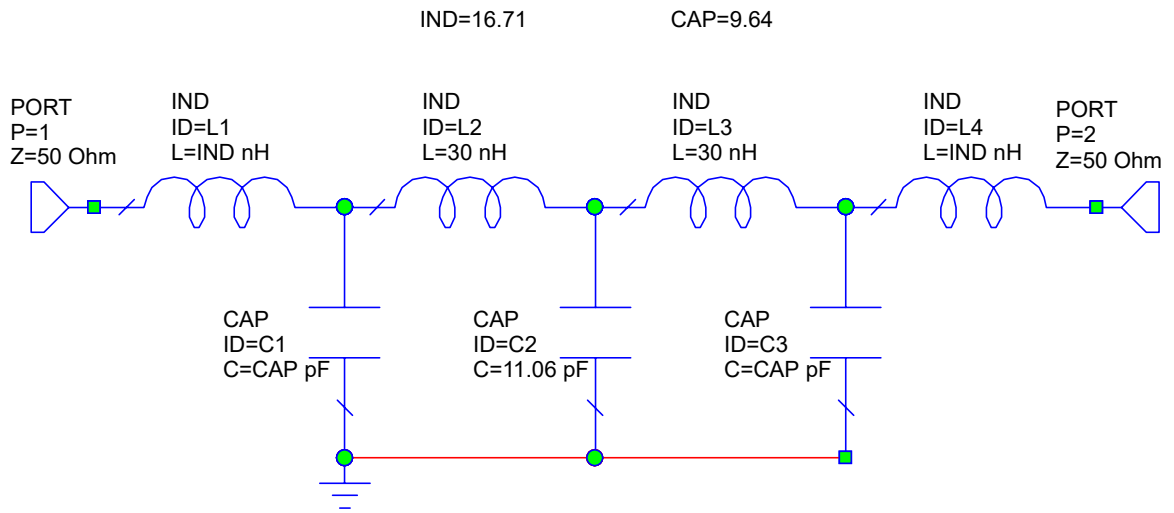


3. Repeat Step 1, then select **lpf:DB(|S(2,1)|)** as the **Measurement**, select **Meas > Goal** as the **Goal Type**, deselect **Max** under **Range**, type "500" as the **Stop** value, type "-1" as the attenuation **Goal** in dB, and then click **OK**.
4. Repeat Step 1, then select **lpf:DB(|S(2,1)|)** as the **Measurement**, select **Meas < Goal** as the **Goal Type**, deselect **Min** under **Range**, type "700" as the **Start** value, type "-30" as the **Goal** in dB, and then click **OK**.

Optimizing the Circuit

To view the optimization in progress, ensure that the graph window is visible before performing these steps.

1. Choose **Simulate > Optimize**. The Optimizer dialog box displays.
2. Under **Optimization Methods** select **Random (Local)** from the drop-down list, type "5000" in **Maximum Iterations**, and then click **Start**. The optimization runs.
3. When the optimization is complete, click the **X** box to exit the Optimizer dialog box. The optimized response in the following schematic and graph should display. Note that your IND and CAP values may vary slightly.



4. Save and close the project.

Chapter 5. Microwave Office: Creating Layouts from Schematics

Layouts are views of the physical representations of a schematic. Layout is a critical part of high-frequency circuit design and simulation, since the response of a circuit is dependent on the geometric shapes with which it is composed.

Layouts in Microwave Office

Microwave Office's layout capability, architected using advanced object-oriented programming techniques, is tightly integrated with its schematic and EM structure design capabilities. The Layout View is another view of the schematic, so any modifications you make to a schematic are simultaneously updated in its corresponding layout. This eliminates the need for complicated design synchronization and back annotation before you perform your simulations.

The following example demonstrates basic layout features. Microwave Office software offers many advanced features that allow you to generate complex layouts such as MMIC circuits and various types of multi-layer boards. For more advanced layout topics, see the [Microwave Office Layout Guide](#).

Layout Tips and Tricks

The following keyboard shortcuts are helpful as you use Microwave Office's layout capability.

Keystrokes	Layout Function
Press the + key	Zoom in
Press the - key	Zoom out
Press the Home key	Full view
Press the Ctrl key, select a shape, move the mouse	Snap to corners, edges, and centers of circles
Select a shape, hold down the mouse button, press the Tab key or Space bar	Move shape with coordinate entry
Press Ctrl + Shift while clicking on layered shapes	Cycle through layered shapes/elements and select them individually

NOTE: The *Quick Reference* document lists additional keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Creating a Layout from a Schematic

This example demonstrates how to use Microwave Office software to create a layout from a schematic. It includes the following main steps:

- Importing a Layer Process File (LPF)
- Editing Database Units and Default Grid Size
- Importing a cell library
- Importing and placing a data file in a schematic
- Changing an element symbol
- Placing microstrip lines for layout
- Assigning an artwork cell to a schematic element
- Viewing a layout

- Anchoring a layout cell
- Creating an artwork cell
- Manipulating the MTRACE2 element in layout
- Snapping functions in layout
- Exporting a layout

Creating a New Project

The example you create in this chapter is available in its complete form as *layout_example.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl-click** the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. You can use this example file as a reference.

To create a project:

1. Choose **File > New Project**.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**layout_example**" as the project name, and then click **Save**.

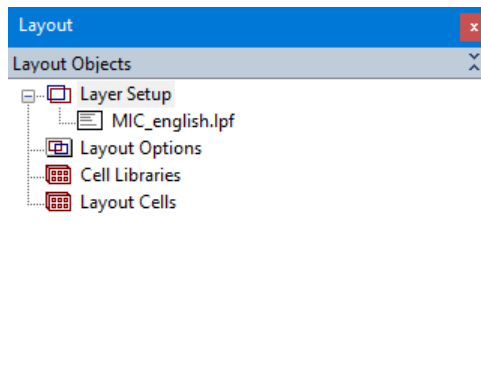
Importing a Layer Process File

A Layer Process File (LPF) defines the default settings for the Layout View, including drawing layers, layer mappings, 3D views, and EM layer mappings.

To import an LPF:

1. Click the **Layout** tab to display the Layout Manager.
2. Right-click **Layer Setup** in the Layout Manager, and choose **Import Process Definition**. The Import Process Definition dialog box displays.
3. Locate the program directory (*C:\Program Files\AWR\AWRDE\14* or *C:\Program Files (x86)\AWR\AWRDE\14* is the default installation directory) and double-click it to open it. If you changed the default installation directory, then locate that directory instead when the program directory is referenced.
4. Select the *MIC_english.lpf* file and click **Open**. Click **Replace** when prompted to replace the default lpf file. This step ensures that your project uses the same project units and drawing layers as this example.

The following figure shows an example Layout Manager.

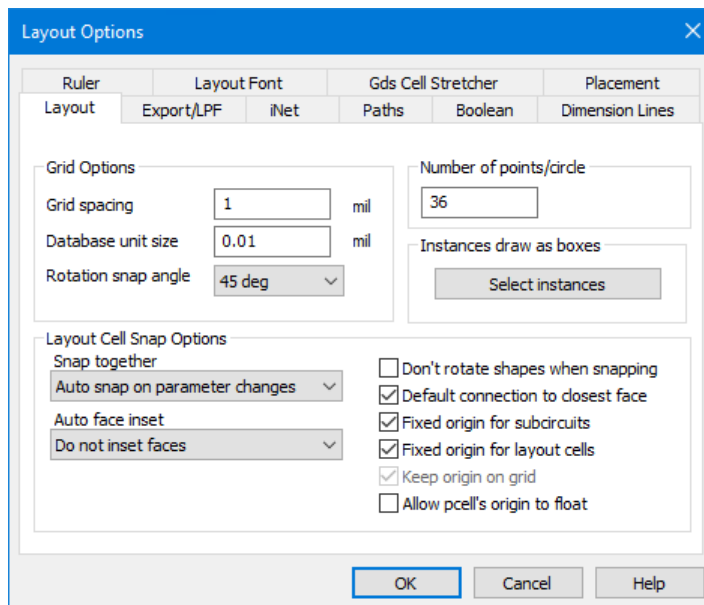


Editing Database Units and Default Grid Size

A database unit is defined as the smallest unit of precision for a layout. It is very important that this parameter is not changed after it is set. Changing database units can cause rounding errors that may lead to problems in the layout file. The grid size is important because many IC designs must reside on a grid. The grid must be greater than or equal to the database unit. Because the grid multiplier's smallest unit is .1x, you should set the grid to 10 times the database unit to prevent having a smaller grid than database unit.

To set the database unit and grid size:

1. Choose **Options > Layout Options**. The Layout Options dialog box displays.
2. On the **Layout** tab, type ".1" in **Grid spacing** and ".01" in **Database unit size**.
3. In **Snap together**, select **Auto snap on parameter changes** and then click **OK**.

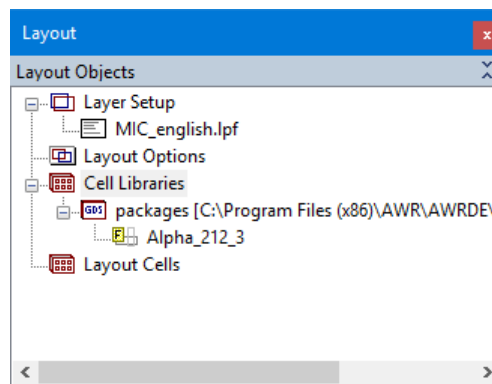


Importing a GDSII Cell Library

Cell libraries are used in Microwave Office software to provide both the physical packages and footprints for printed circuit board or hybrid design processes, as well as the standard artwork cells used in MMIC and RFIC design processes. Microwave Office software supports the GDSII file format as the native drawing tool format.

To import a GDSII cell library:

1. Right-click **Cell Libraries** in the Layout Manager and choose **Import GDSII Library**.
2. Browse to the *C:\Program Files\AWR\AWRDE\14\Examples* or *C:\Program Files (x86)\AWR\AWRDE\14\Examples* directory and double-click it to open it.
3. Select the *packages.gds* file and click **Open**. The imported cell library displays in the Layout Manager. If a warning message displays, click **OK**.



Importing a Data File

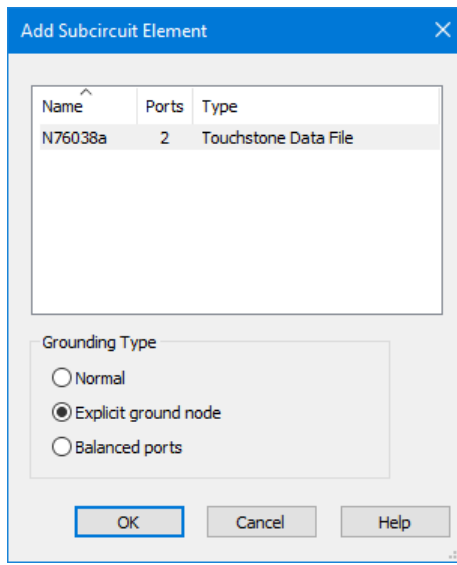
To import a data file:

1. In the Project Browser, right-click **Data Files** and choose **Import Data File**. The Browse for File dialog box displays.
2. Browse to the *C:\Program Files\AWR\AWRDE\14\Examples* or *C:\Program Files (x86)\AWR\AWRDE\14\Examples* directory and double-click it to open it.
3. Select the *N76038a.s2p* file and then click **Open**.

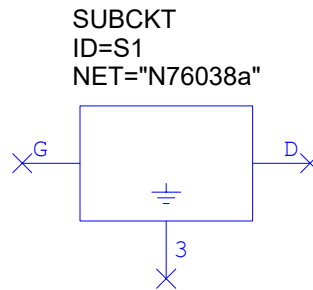
Placing a Data File in a Schematic and Adding a Ground Node

To place a data file in a schematic and add a ground node:

1. Right-click **Circuit Schematics** in the Project Browser, choose **New Schematic**, and create a schematic named "**qs layout**", then click **Create**.
2. Choose **Draw > Add Subcircuit**, or click the **SUB** button on the toolbar to add a subcircuit. The Add Subcircuit Element dialog box displays as shown in the following figure.
3. Select **N76038a** from the list and select **Explicit ground node** as the **Grounding type**.



4. Click **OK** to place it in the schematic window.

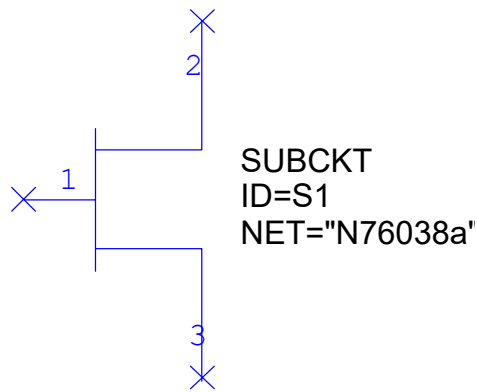


Changing the Element Symbol

You can change the subcircuit symbol to represent a FET so that you can see which nodes correspond to the gate, drain, and source.

To change the element symbol:

1. Double-click the subcircuit element in the schematic window. The Element Options dialog box displays.
2. Click the **Symbol** tab.
3. Select **FET@system.syf** in the list box, and then click **OK**.



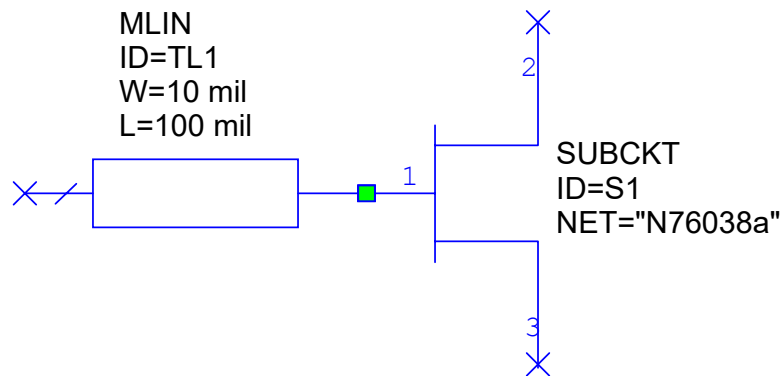
Placing Microstrip Elements for Layout

Microstrip elements have default layout cells associated with each element. The layout cells are parameterized and dynamically sized to the values specified for each parameter.

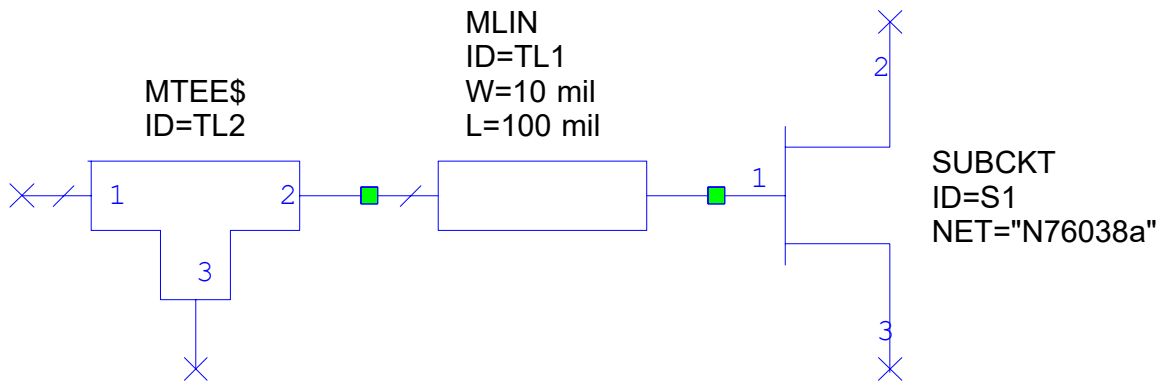
Microwave Office software has specialized microstrip elements called iCells (intelligent cells) that do not require any parameter values for the dimensions of the element. iCells automatically inherit the necessary parameters from the connecting element.

To place microstrip elements:

1. In the Element Browser, expand the **Microstrip** category, then click the **Lines** group. Select the MLIN model and place it onto node 1 of the N7068a subcircuit in the schematic window. Change its parameters as shown in the following figure.

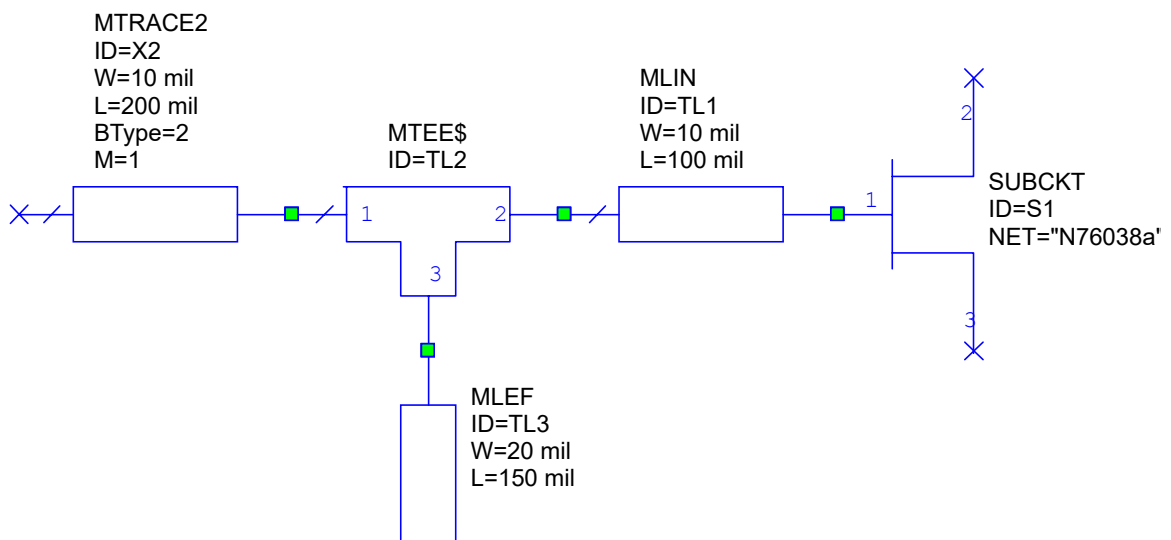


2. In the **Microstrip** category, click the **Junctions** group. Select the MTEES\$ model and place it in the schematic window connected to the MLIN element as shown in the following figure.

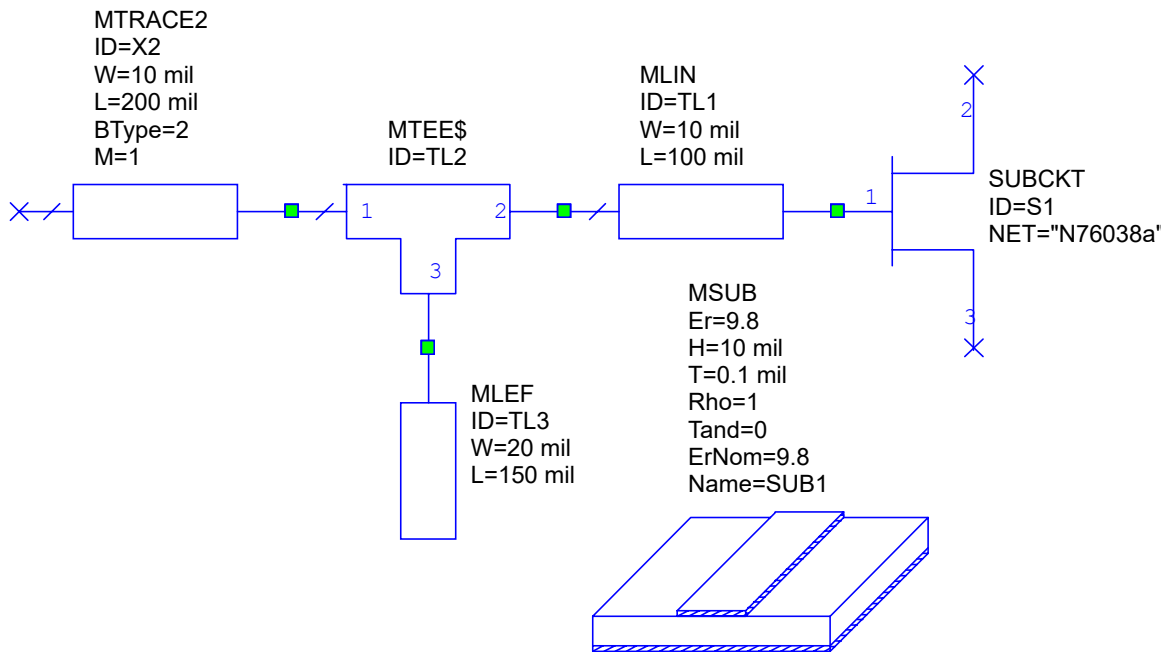


NOTE: Elements with names ending in "\$" inherit their attributes from the ports to which they connect. Elements with names ending in "X" are created from a lookup table of EM-generated models. Thus, the name "MTEEX\$" is a microstrip tee junction based on an EM model lookup table that inherits its widths from the ports to which it connects.

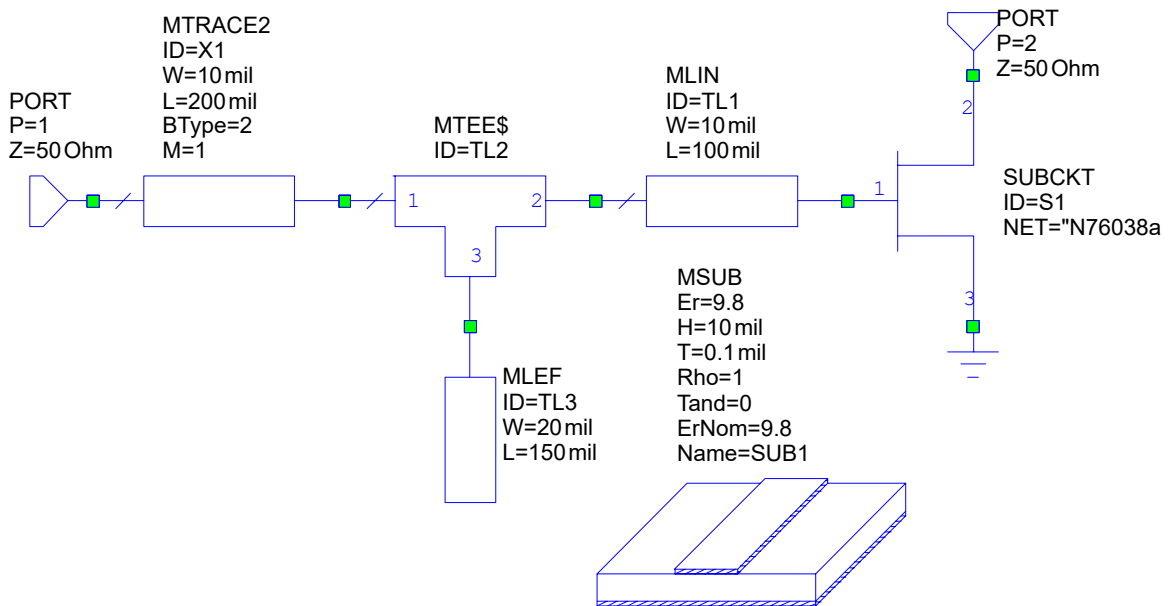
3. In the **Microstrip** category, click the **Lines** group. Select the MTRACE2 model and place it in the schematic window onto node 1 of the MTEES\$ element.
4. Select the MLEF model in the same group and drag it into the schematic window. Right-click three times to rotate the element, then position it onto node 3 of the MTEES\$ element.
5. Double-click the MTRACE2 element in the schematic window to display the Element Options dialog box.
6. On the **Parameters** tab, edit the MTRACE2 parameters to match those shown in the following figure, then click **OK**.



7. Repeat steps 5 and 6 for the MLEF elements to match their parameters to those shown in the previous figure.
8. Click the **Substrates** category, then select the MSUB model and place it on the schematic window as shown in the following figure.
9. Double-click the MSUB element in the schematic window to display the Element Options dialog box. On the **Parameters** tab, edit the MSUB parameters to match those shown in the following figure, then click **OK**.



10. Choose **Draw > Add Port**, or click the **Port** button on the toolbar, move the cursor onto the schematic, position the port on the left node of the MTRACE2 element as shown in the following figure, and click again to place it.
11. Add another port to node 2 of the SUBCKT element. Right-click three times to rotate the port, position it, and click again to place it.
12. To complete the schematic, click the **Ground** button on the toolbar, move the cursor into the schematic, position the ground on node 3 of the SUBCKT element, and click again to place it.

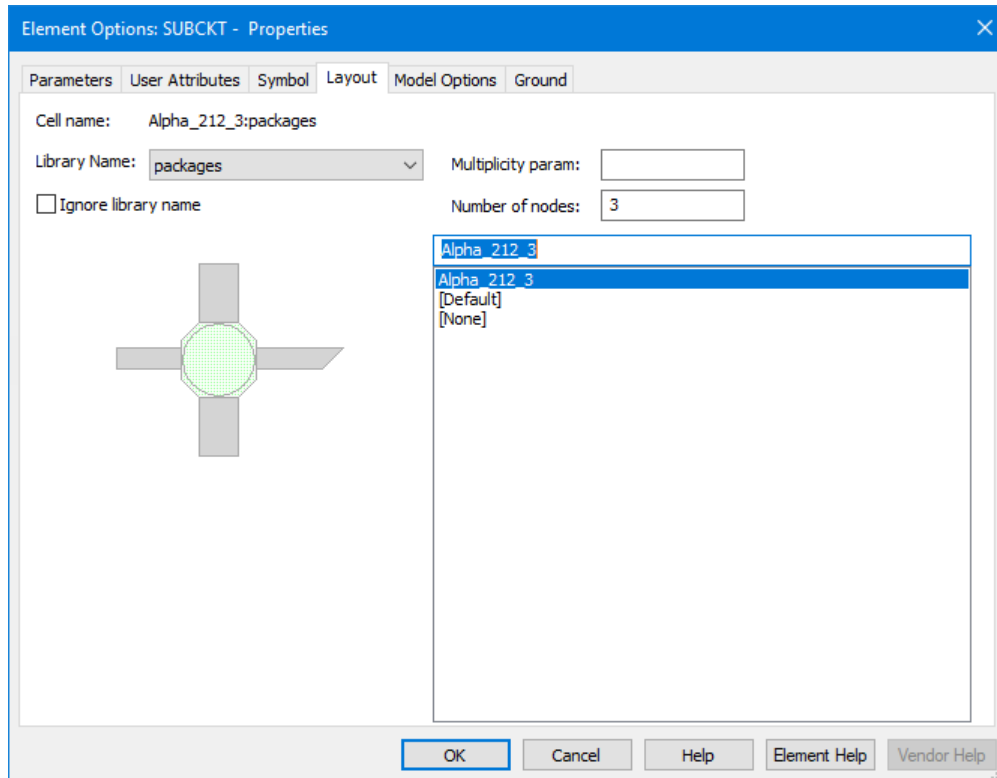


Assigning an Artwork Cell to a Schematic Element

Artwork cells that represent a package layout can be assigned to a schematic element.

To assign an artwork cell:

1. Double-click the "N76038a" subcircuit element in the schematic window to display the Element Options dialog box.
2. Click the **Layout** tab.
3. Select **packages** as **Library Name** and **Alpha_212_3** in the cell list at the right of the dialog box, then click **OK**.



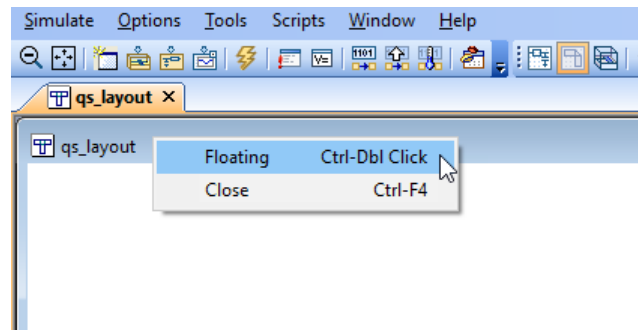
Viewing a Layout

The schematic and layout are different views of the same database. Edits made to the parameters in the schematic are instantly updated in the layout, and edits made in the layout are instantly updated in the schematic.

To view a layout:

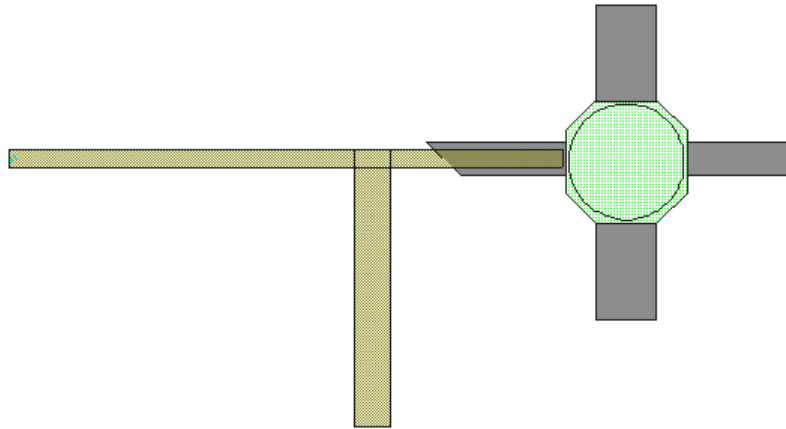
1. Click the schematic window to make it active.
2. Choose **View > View Layout** or click the **View Layout** button on the toolbar to view a layout representation. The layout displays in a layout window.

NOTE: To maximize work area you can "float" a layout window by right-clicking its title bar and choosing **Floating**.



Snapping Layout

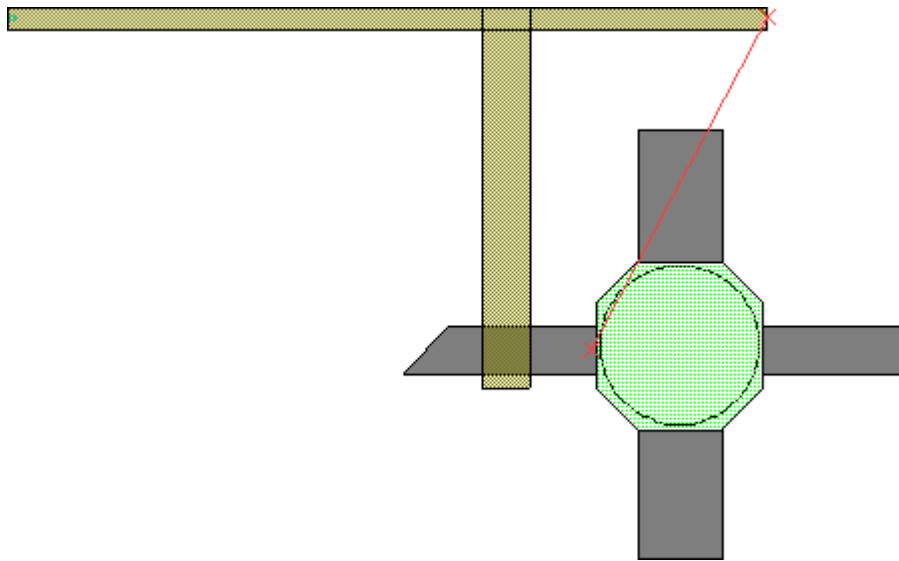
1. Choose **Edit > Select All** to select all of the layout cells.
2. Choose **Edit > Snap Objects > Snap Together** to snap all of the layout cells faces together at once.



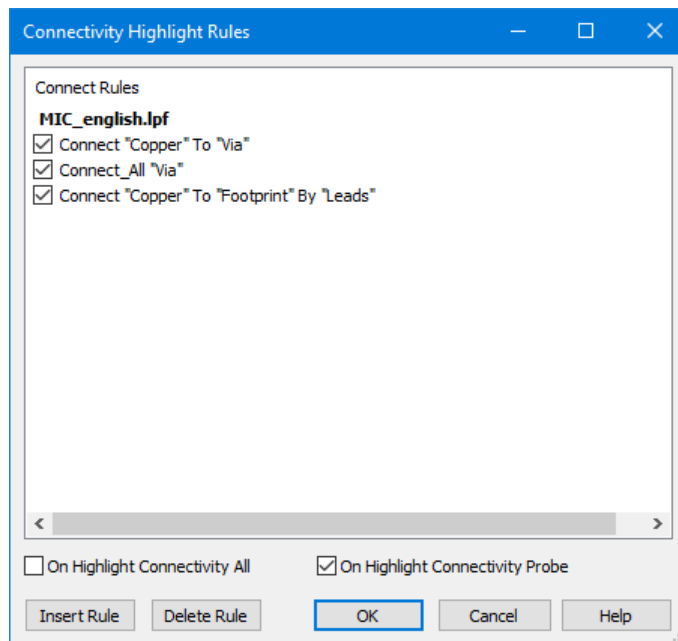
Running the Connectivity Checker

To run the Connectivity Checker:

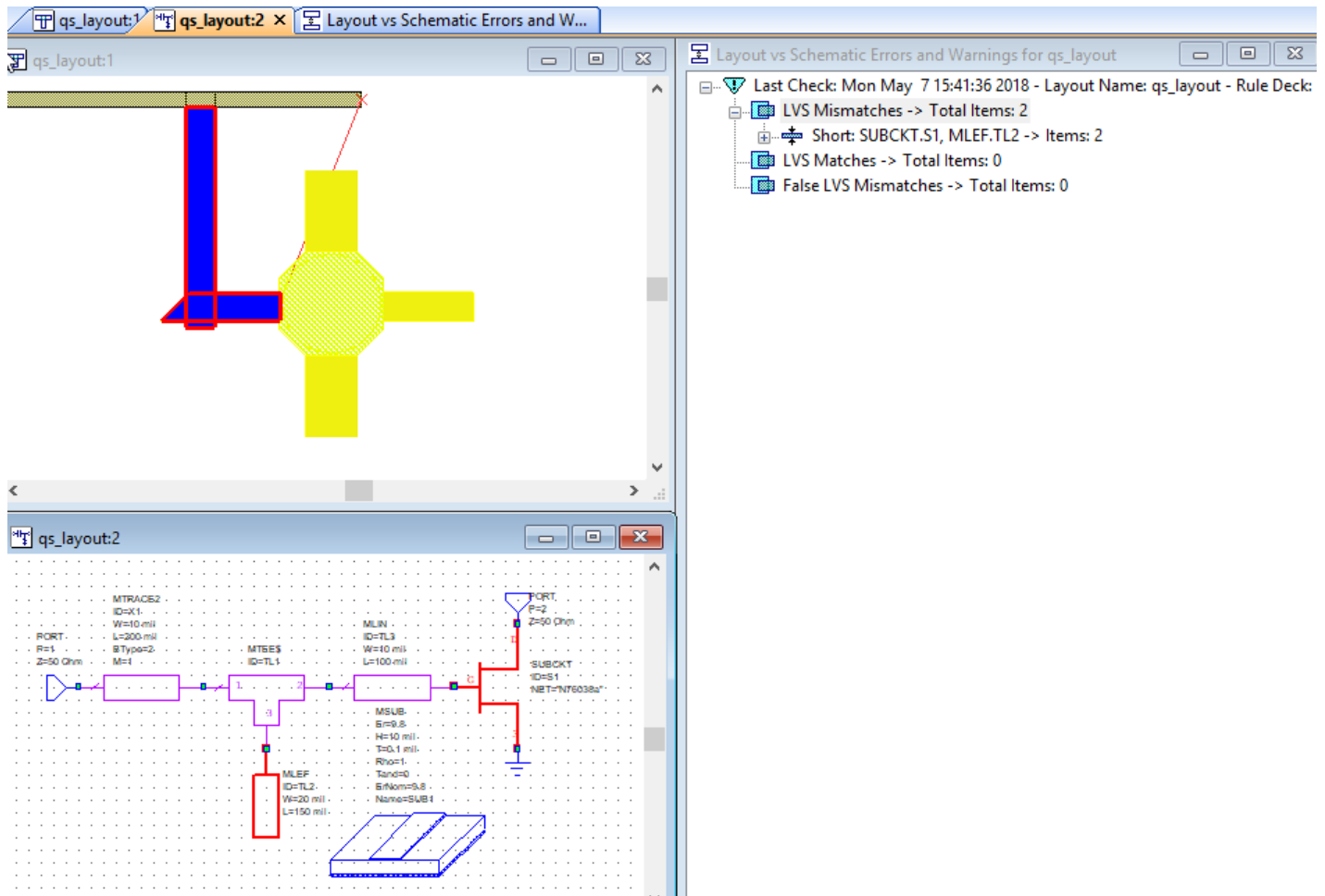
1. If your layout displays differently, click and drag the "Alpha_212_3" cell in the layout window until it is positioned as shown in the following figure. To rotate the cell, select it and then right-click and choose **Rotate**. When the cursor displays as a 90-degree arc, hold down the mouse button and move the cursor clockwise in 45-degree increments until the cell is properly oriented.



2. Choose **Verify > Highlight Connectivity Rules**. In the Connectivity Highlight Rules dialog box, select the rules and options shown in the following figure, then click **OK**.



3. Choose **Verify > Run Connectivity Check**. An LVS errors window opens and lists any violations. If you select a violation, the corresponding elements are highlighted in both the schematic and layout windows, as shown in the following figure.



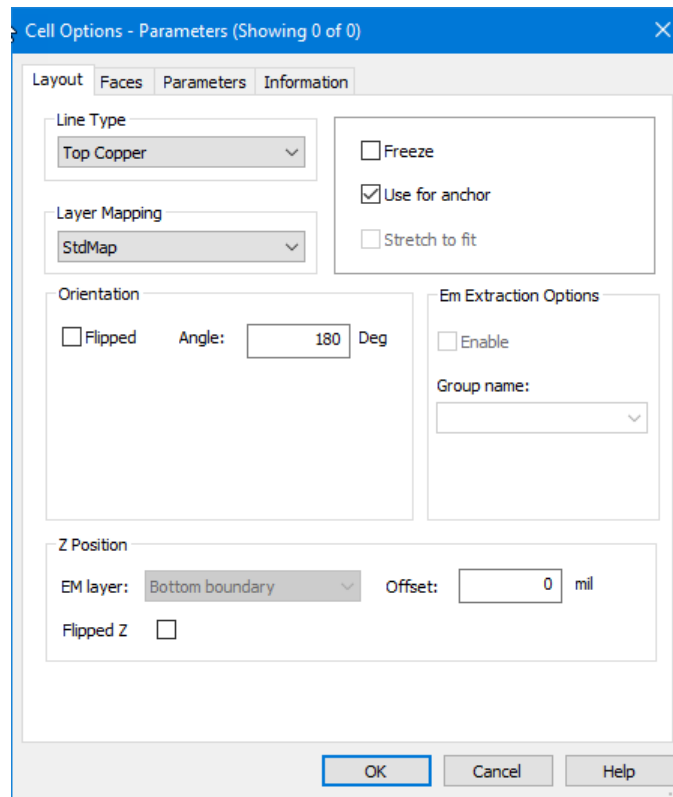
4. Choose **Verify > Clear LVS Errors**.
5. Choose **Edit > Select All** to select all of the layout cells.
6. Choose **Edit > Snap Objects > Snap Together** to snap the layout cells again.
7. Choose **Verify > Run Connectivity Check**. Notice that there are no errors listed in the LVS errors window.

Anchoring a Layout Cell

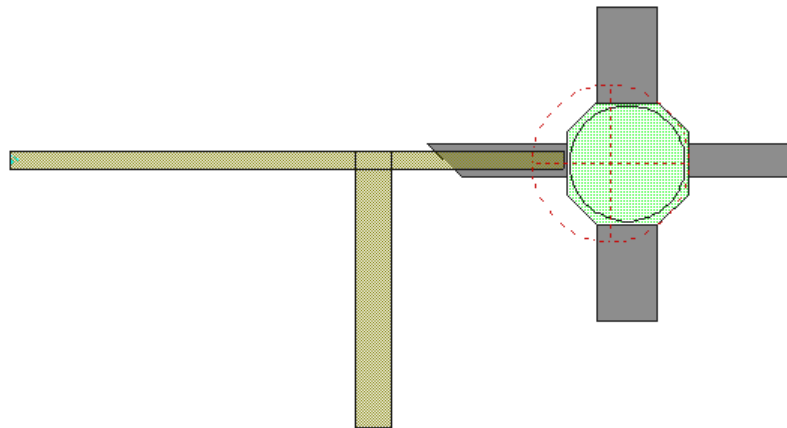
Layout cells have various properties that determine the connectivity of each cell in the Layout View. One of the important properties is anchoring. Anchoring a layout cell holds the cell in place so that it cannot be moved by snapping functions. An anchored layout cell is typically used to define a reference point for the layout.

To anchor a layout cell:

1. In the layout window, select the "Alpha_212_3" artwork cell. Right-click and choose **Shape Properties** to display the Cell Options dialog box. You may have all the layout shapes selected from previous operations, so make sure to deselect all the element first.



2. Click the **Layout** tab, select the **Use for anchor** check box, and click **OK**. The artwork cell now has an anchor symbol as shown in the following figure.

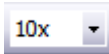


Creating an Artwork Cell

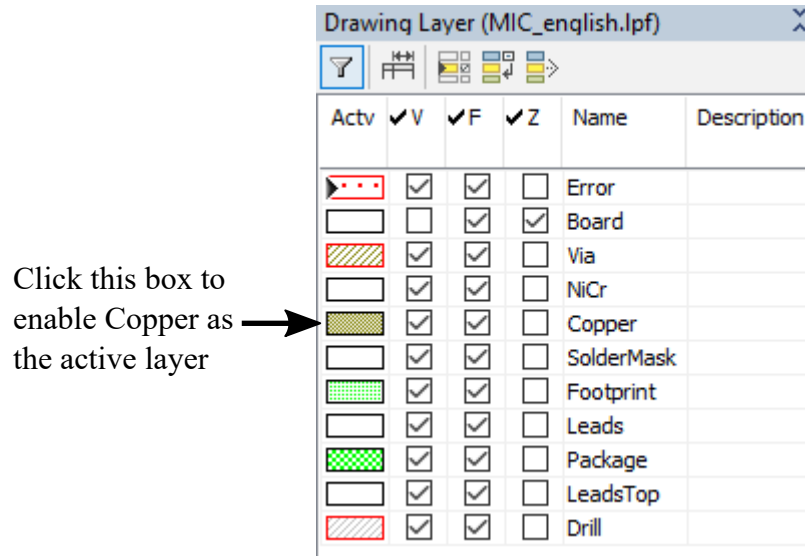
To create an artwork cell:

1. Click the **Layout** tab to open the Layout Manager.
2. Right-click **packages** under **Cell Libraries** and choose **New Layout Cell**. The Create New Layout Cell dialog box displays.

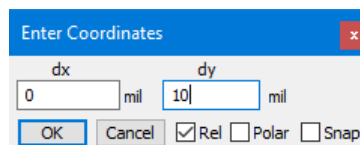
3. Name the cell "**chip cap**" and click **OK**. A drawing window displays in the workspace.
4. Click the **Grid Spacing** button on the toolbar and set it to 10x. (Ensure that the Schematic Layout toolbar is displayed by right-clicking on the toolbar and selecting **Schematic Layout**.)



5. Click **chip cap** under **packages** in the upper Layout Manager window, and then click the Copper box in the **Actv** column of the Drawing Layer pane to enable Copper as the active layer, as shown in the following figure. (Do not select any check boxes, as it changes other drawing layer properties.)



6. Choose **Draw > Rectangle**.
7. Move the cursor into the drawing window and then press the **Tab** key or **Space** bar to display the Enter Coordinates dialog box.
8. Type the values "**0**" and "**10**" in **x** and **y**, respectively, and click **OK**.

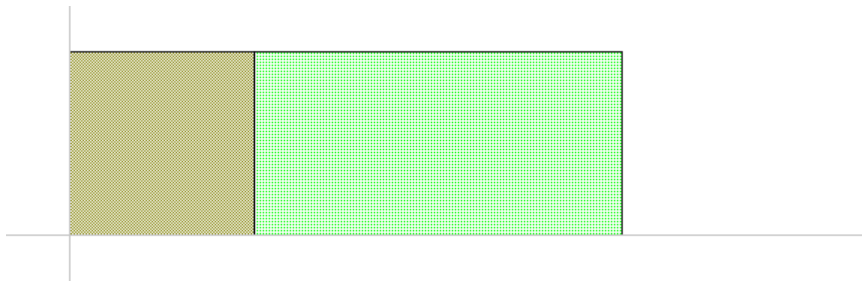


NOTE: Coordinate entry allows you to type in coordinate values; you can also click and drag to draw shapes.

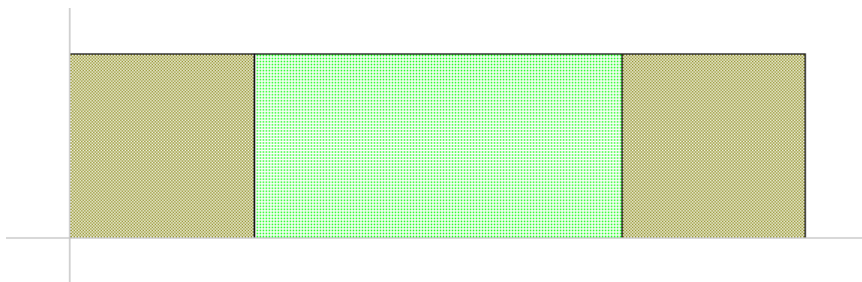
9. Press the **Tab** key or **Space** bar again to display the Enter Coordinates dialog box.
10. Type the values "**10**" and "**-10**" in **dx** and **dy**, respectively, and click **OK**. (**Rel** is automatically selected.) The following figure shows the resulting drawing. (You may need to adjust your view by choosing **View > Zoom In** or **View > Zoom Out**.)



11. Click the Footprint box in the left column of the lower pane of the Layout Manager to enable Footprint as the active layer.
12. Click the "chip cap" window to make it active.
13. Choose **Draw > Rectangle**.
14. Move the cursor into the "chip cap" window, then press the **Tab** key or **Space** bar to display the Enter Coordinates dialog box.
15. Type the values "**10**" and "**10**" in **x** and **y**, respectively, and click **OK**.
16. Press the **Tab** key or **Space** bar again to display the Enter Coordinates dialog box.
17. Type the values "**20**" and "**-10**" in **dx** and **dy**, respectively, and click **OK**. The following figure shows the resulting drawing.



18. Click the copper square in the "chip cap" window, and press **Ctrl + C** then **Ctrl + V** to copy and paste it. Slide the mouse to position the copied square along the right edge of the rectangle as shown in the following figure, and click to place it.

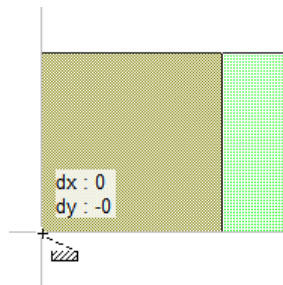


Add Ports to an Artwork Cell

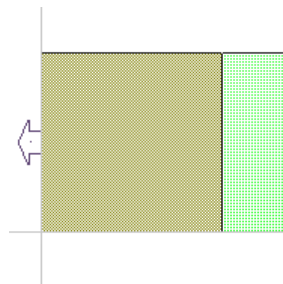
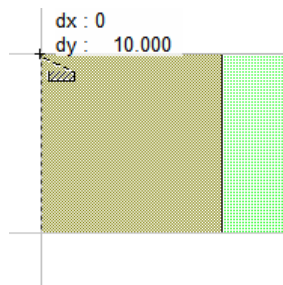
Ports in the Artwork Cell Editor define the faces to which other layout cells connect. The orientation of the port arrow determines the direction of connection to the adjacent layout cell.

To add ports to an artwork cell:

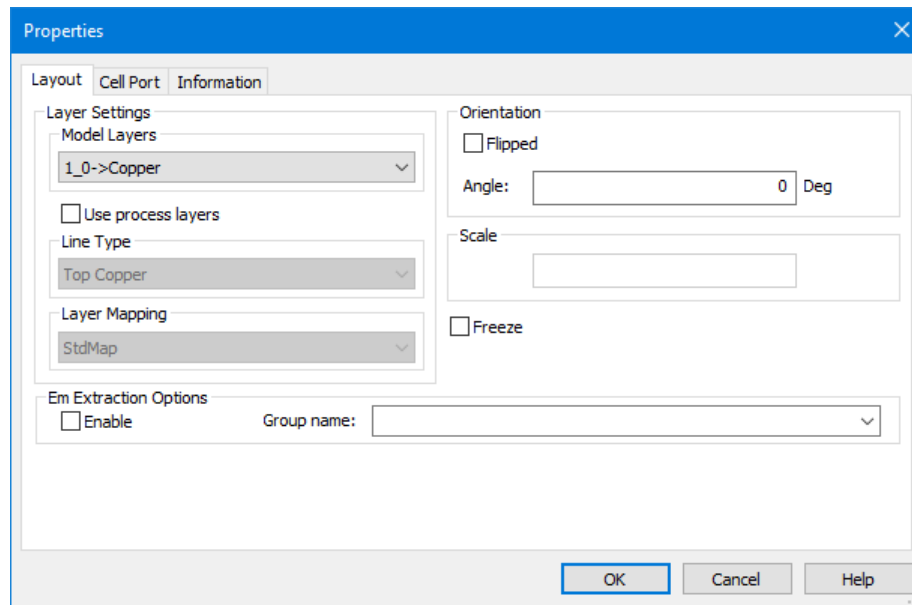
1. Choose **Draw > Cell Port**.
2. Move the cursor into the "chip cap" window. To create gravity points for positioning and aligning shapes in the layout, press and hold the **Ctrl** key while you move the cursor over the bottom left vertex of the square until a diamond symbol displays on the vertex. Do not release the **Ctrl** key.



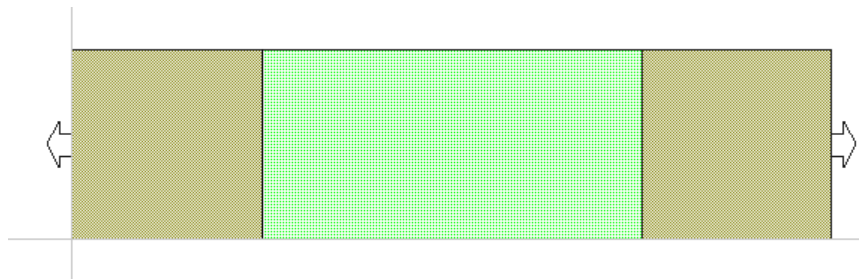
3. With the **Ctrl** key still pressed, click and hold down the mouse button while moving the cursor to the top vertex, until another square displays on that vertex. Release the mouse button and the **Ctrl** key.



4. To successfully run the Connectivity Checker, you must correctly set the port properties of the artwork cell. Select cell port 1, then right-click and choose **Shape Properties**.
5. In the Properties dialog box, click the **Layout** tab and in **Model Layers** select **1_0->Copper**, then click **OK**.



- Repeat these steps to place a port on the opposite side of the drawing, starting at the top vertex and drawing down.

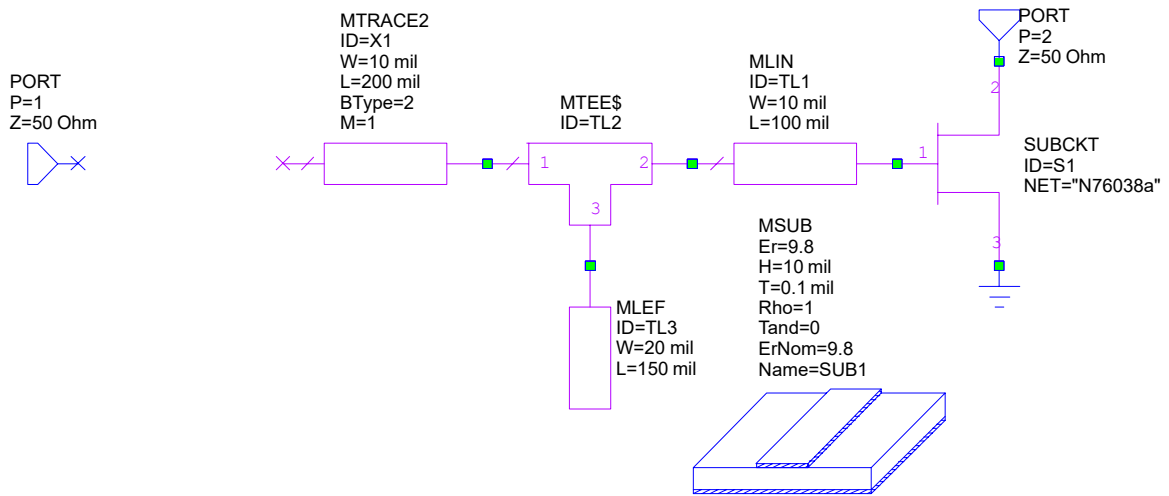


- Click the **X** at the top right of the "chip cap" window and click **Yes** when prompted to save the cell edits.

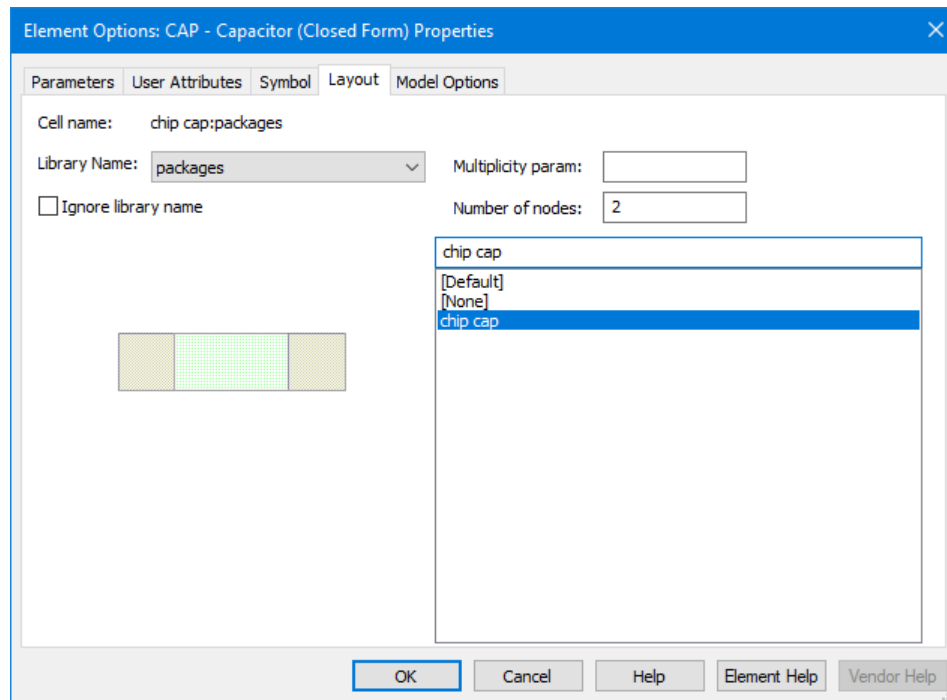
Editing the Schematic and Assigning a Chip Cap Cell

To edit the schematic and assign a chip cap cell:

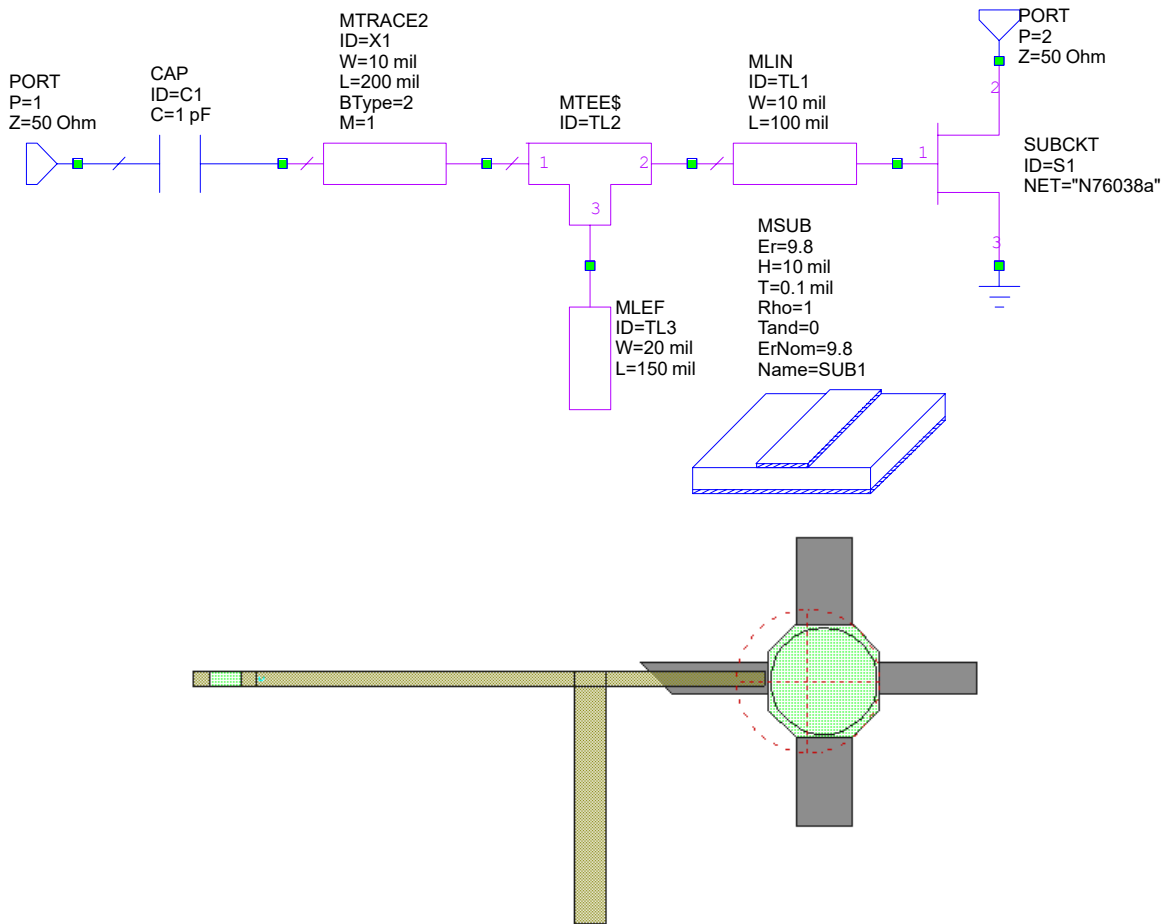
- Click PORT 1 in the schematic window.
- Press and hold the **Ctrl** key while you drag the port away from the MTRACE2 element, as shown in the following figure.



3. In the Element Browser, expand the **Lumped Element** category, then click the **Capacitor** group. Select the CAP model, and place it in the schematic window between PORT 1 and the MTRACE2 element.
4. Double-click the CAP C1 element in the schematic window. The Element Options dialog box displays.
5. Click the **Layout** tab.
6. Select **packages** as **Library Name** then select "chip cap" from the list of cells, then click **OK**.



7. Choose **View > View Layout**. The new layout displays in the workspace. Choose **Edit > Select All**, and then **Edit > Snap Objects > Snap Together** to snap the layout together. The layout and corresponding schematic are shown in the following figures.

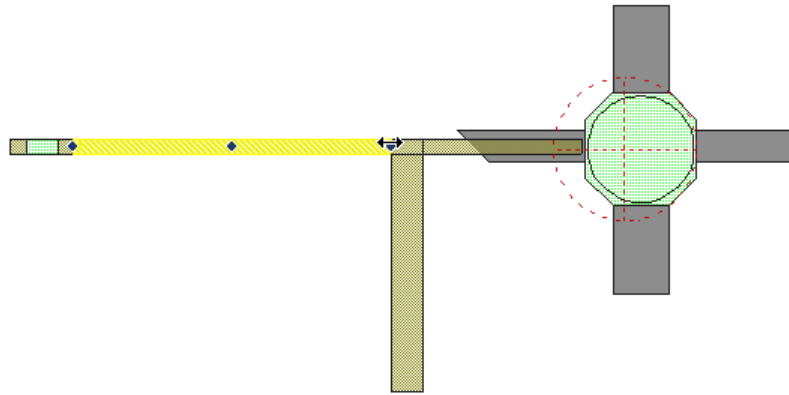


Routing the MTRACE2 Element in Layout

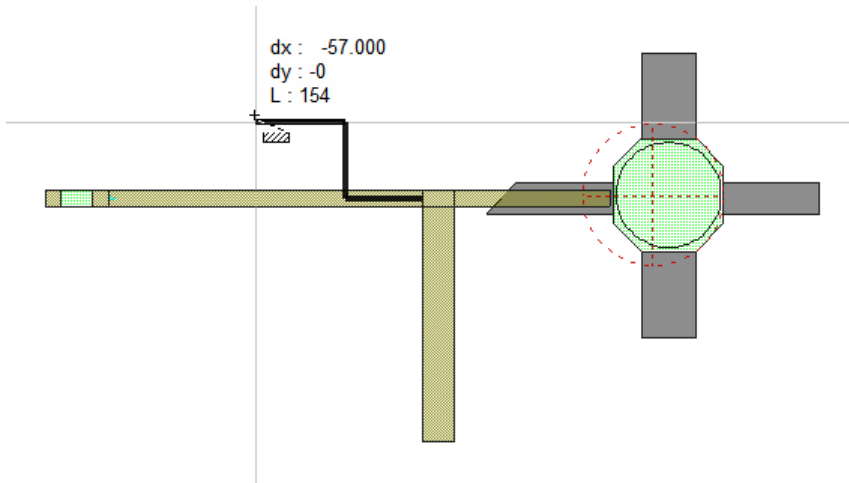
The MTRACE2 element is a special element that you can edit in the Layout View to route a microstrip line.

To route the MTRACE2 element:

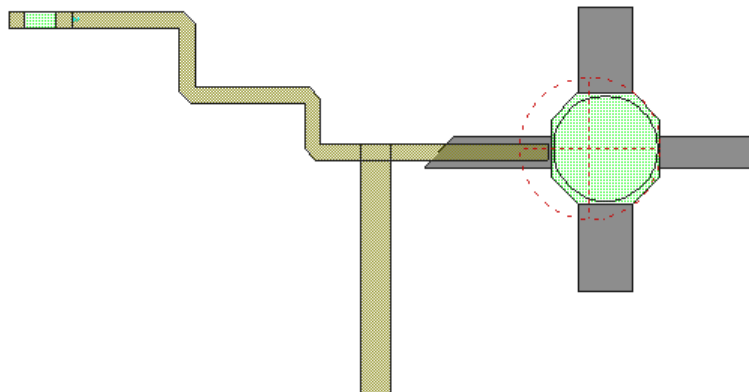
1. Double-click the MTRACE2 element in the Layout View to activate the blue grab diamonds.
2. Move the cursor over the right-most diamond until a double arrow symbol displays as shown and double-click it to activate the routing tool.



3. Move the routing tool to another point in the direction of MTRACE2 as shown, and click to place. (Right-click to delete the last point; press the **Esc** key to cancel the activity). If you move the routing tool in the opposite direction of MTRACE2 then the final route will be flipped.



4. Continue to route points by moving the routing tool and clicking to place, then double-click to complete the routing.



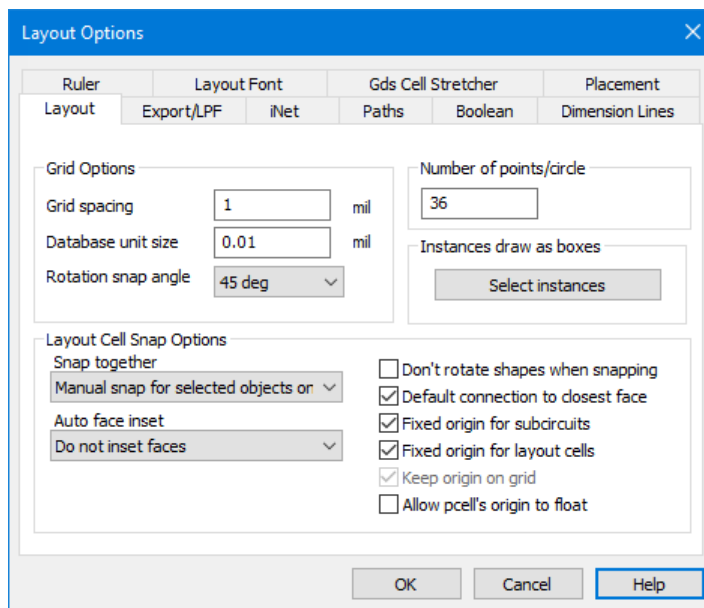
NOTE: MLIN is a straight element with a width you can change in the layout. You can edit the MTRACE2 elements in the layout to create jogs and bends and chamfered corners. You can edit the MCTRACE element to create jogs and bends with rounded corners.

Snapping Functions for Layout Cells

Snapping functions connect the faces of artwork cells in various configurations. You can set snapping options from the Layout Options dialog box.

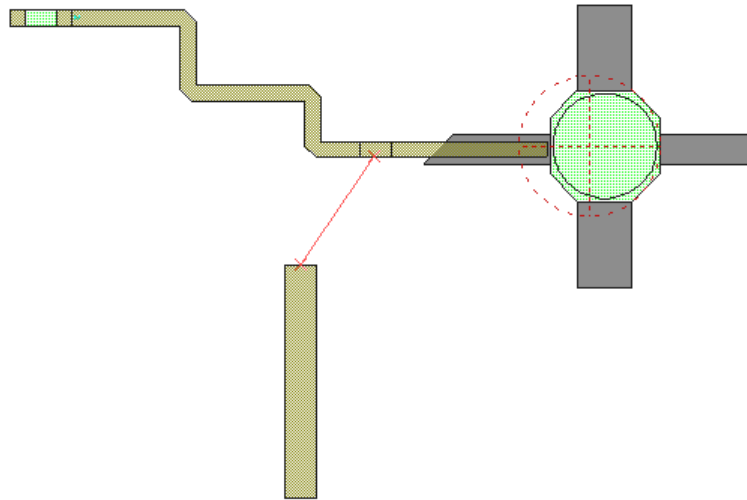
To specify the snapping options:

1. Choose **Options > Layout Options**. The Layout Options dialog box displays.
2. On the **Layout** tab, select **Manual snap for selected objects only** under **Snap Together**, then click **OK**.

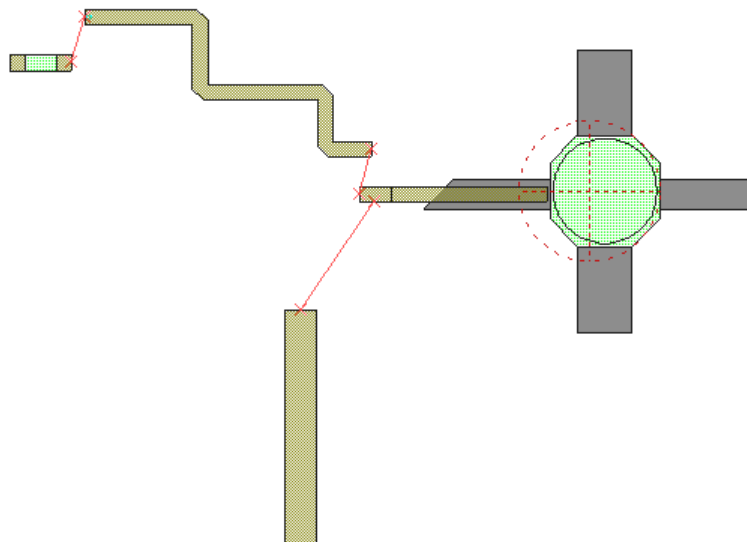


To separate the layout cells so the change in snapping options is viewable:

1. Click the MLEF layout cell and drag it to a new position as shown in the following figure.



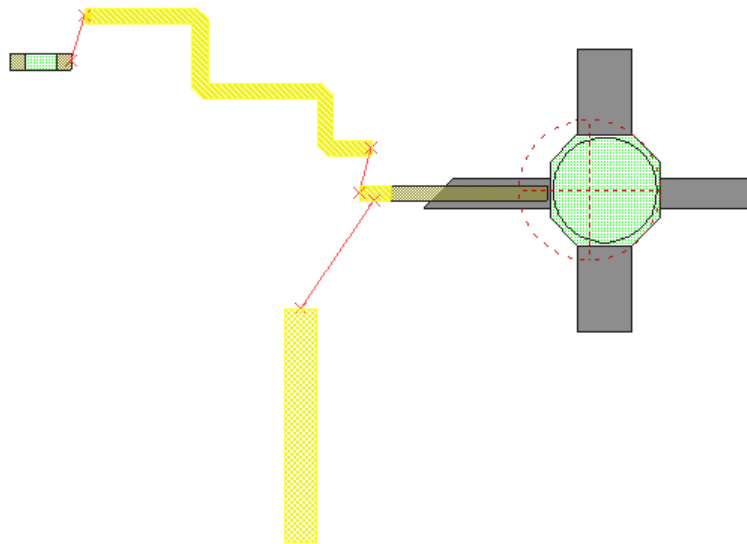
2. Repeat step 1 with the MTRACE2 element and the chip cap cell. Position the layout cells as shown in the following figure.



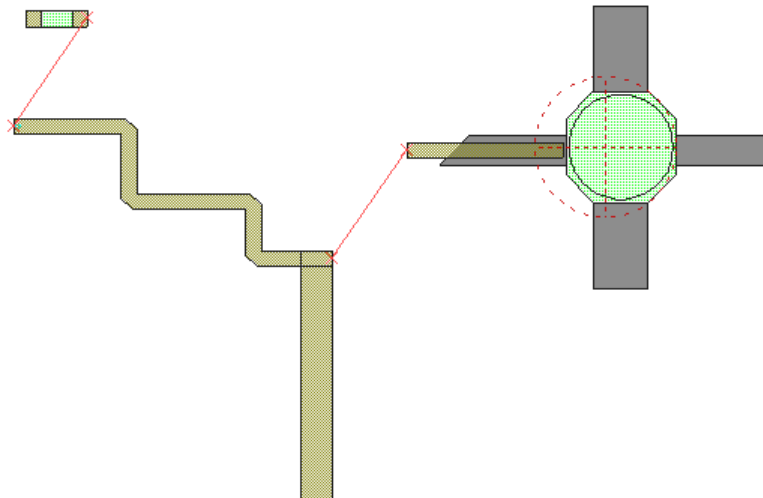
The red lines indicate that the faces of the layout cells are not snapped together.

To snap a selected set of layout cells together:

1. Hold down the **Shift** key and select the MLEF, MTRACE2, and MTEES\$ layout cells in the layout window. The first cell selected serves as the anchor.

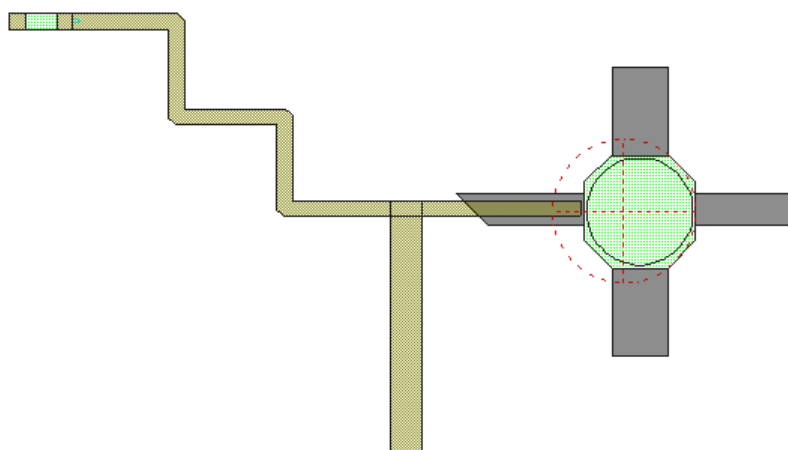


2. Choose **Edit > Snap Objects > Snap Together**, or click the **Snap Together** button on the toolbar. Note that the chip cap layout cell and MLIN layout cell are not snapped together.



To snap all of the faces together:

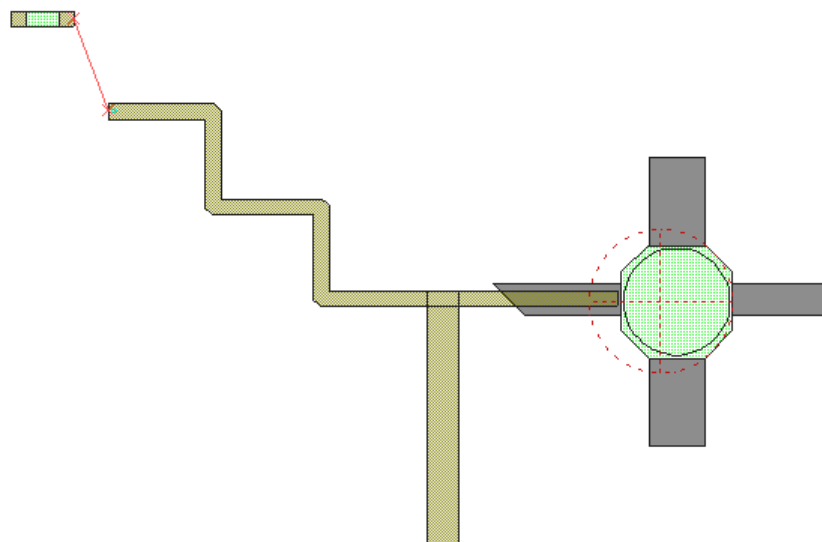
1. Press **Ctrl + A** to select all of the layout cells.
2. Click the **Snap Together** button on the toolbar. The layout displays as shown in the following figure.



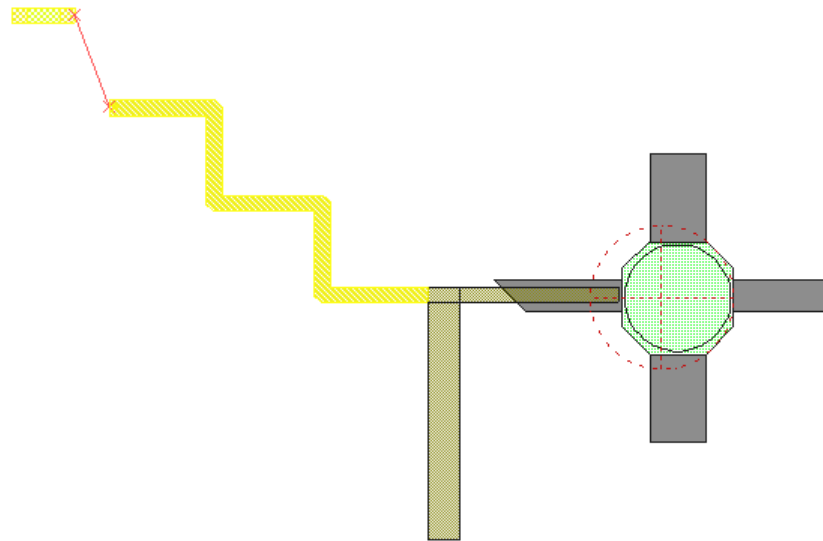
The "snap to fit" function finishes the routing of an MTRACE2 layout cell to a specified adjacent layout cell. In this example, the chip cap layout cell is moved and MTRACE2 re-routes to snap to the chip cap face.

To "snap to fit" MTRACE2 to the chip cap:

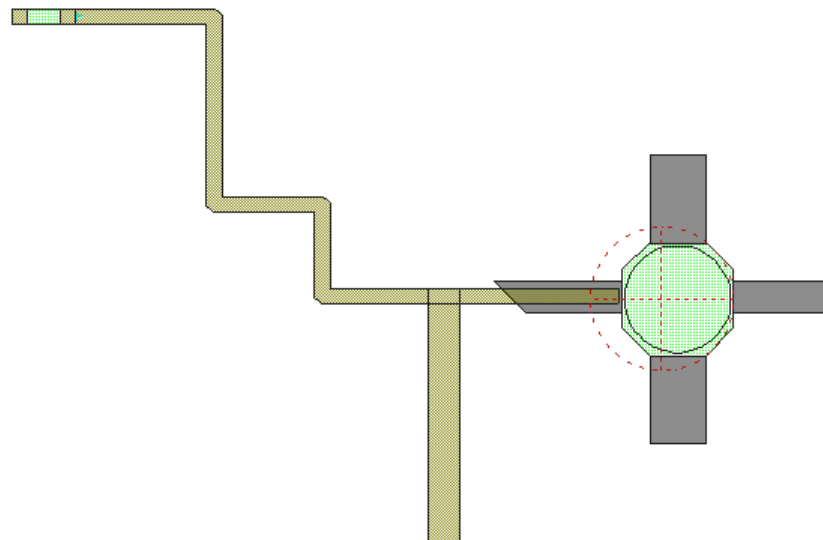
1. Position the chip cap artwork cell as shown in the following figure.



2. Select the MTRACE2 layout cell.



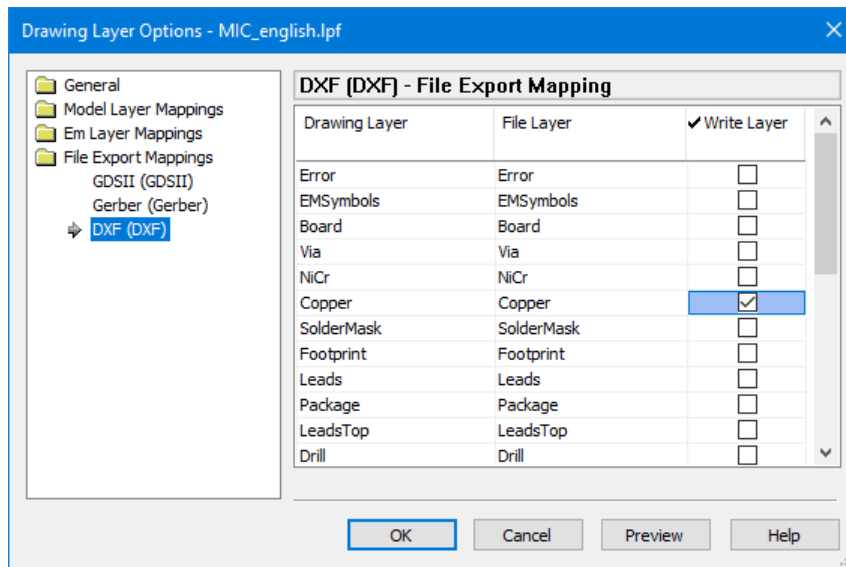
3. Choose **Edit > Snap to fit**, or click the **Snap to Fit** button on the toolbar. The MTRACE2 routes to snap to the chip cap artwork cell as shown in the following figure.



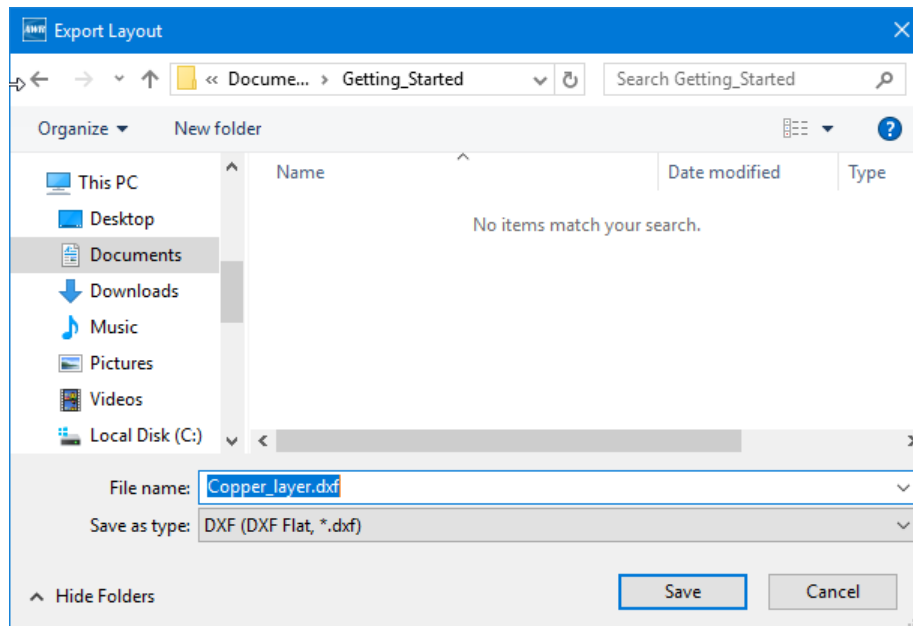
Exporting the Layout

This example exports a DXF file; the NI AWR Design Environment software can also export GDSII and Gerber files. To export a layout:

1. Choose **Options > Drawing Layers** to specify the file layers to export. The Drawing Layer Options dialog box displays.
2. Click the **File Export Mappings** folder.
3. Click **DXF** in the left window, and in the **Write Layer** column, deselect all of the drawing layers except the Copper layer. In the **File Layer** column, name the Copper layer "Copper", then click **OK**.



4. Choose **Layout > Export Layout**. The Export Layout dialog box displays.
5. Select **DXF (DXF Flat,*.dxf)** in **Save As Type**.
6. Type "**CopperLayer**" as the **Filename**, and click **Save** to export the copper file layer to a DXF file.

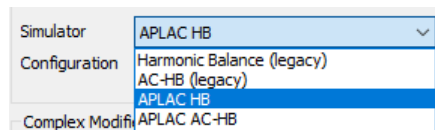


7. Save and close the project.

Chapter 6. Microwave Office: Using the Nonlinear Simulator

Harmonic balance (HB) is an efficient method for the steady-state analysis of nonlinear circuits such as power amplifiers, mixers, multipliers and oscillators. This chapter presents an overview of HB simulations in Microwave Office software. For more detailed information, see “[Harmonic Balance Analysis](#)” in *Simulation and Analysis Guide*.

The NI AWR Design Environment suite includes both the original HB simulator and the APLAC HB simulator. NI AWR highly recommends using the APLAC HB simulator, which is replacing the original HB simulator. The APLAC simulator supports all but a few nonlinear models, so the original HB simulator is still included in the NI AWR Design Environment suite. The APLAC simulator continues to improve every release, while the original HB simulator is no longer being developed. The following exercises use the APLAC **Simulator** as specified in the Add/Modify Measurement dialog box.



Harmonic Balance in Microwave Office

Microwave Office software simplifies HB simulation setup. The schematic entry, measurements, and analysis are accomplished much as they are for a linear simulation. Unlike linear analysis, HB requires the presence of voltage, current, or power sources. Microwave Office software includes a variety of intuitive source elements and allows you to specify single- and multi-tone excitations. The HB simulator is automatically invoked when appropriate excitations are present in the schematic, and corresponding measurements exist.

Single-Tone Analysis

A single-tone HB analysis simulates the circuit at a fundamental frequency, at integer multiples of the fundamental frequency, and at DC. Single-tone harmonic balance requires the specification of a fundamental frequency (or a frequency sweep) and the total number of harmonics.

Multi-Tone Analysis

Multi-tone simulations are used to determine the output of a circuit excited by two or more frequencies that cannot be expressed as integer multiples of one another. Typical examples include the LO and RF signals in a mixer and closely spaced tones used for intermodulation testing of amplifiers.

Nonlinear Measurements

Microwave Office software offers a large number of post-processing functions for viewing simulation results in both the frequency and the time domain. Examples include large signal S-parameters, voltages and currents at arbitrary nodes, intercept points, power spectra and power-added efficiency. Results may be swept over any number of arbitrary parameters.

The following example illustrates some of the key features of the Microwave Office nonlinear simulator.

Creating a Power Amplifier Circuit

This example demonstrates how to use Microwave Office software to simulate a power amplifier circuit using the HB nonlinear simulator.

The basic procedures in this example include:

- Using nonlinear models from the element library
- Creating an IV curve measurement
- Biasing the transistor and measuring voltages and currents
- Adding schematic back annotation
- Importing input/output match schematic
- Creating a hierarchical circuit using subcircuits
- Creating a power out versus frequency measurement
- Creating a dynamic load line measurement
- Adding a two-tone excitation port

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Creating a New Project

The example you create in this chapter is available in its complete form as *nonlinear_example.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl-click** the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. You can use this example file as a reference.

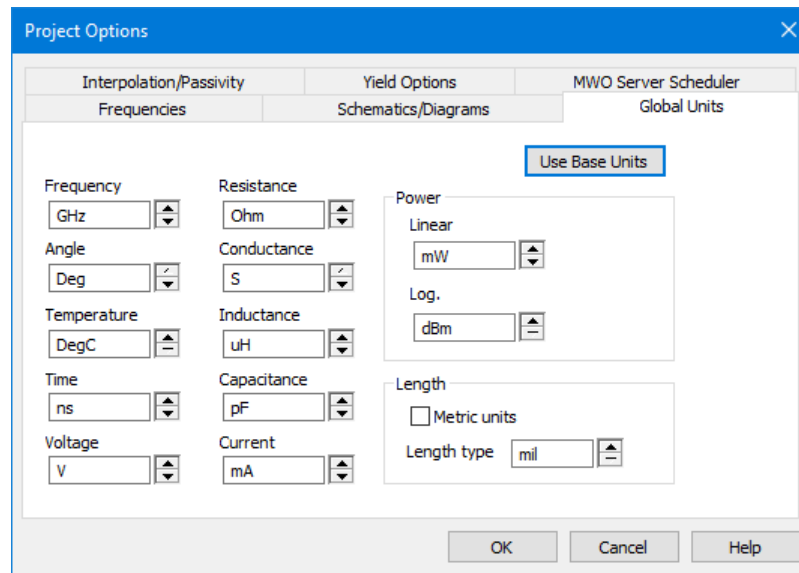
To create a project:

1. Choose **File > New Project**.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**nonlinear_example**" as the project name, and then click **Save**.

Setting Default Project Units

To set default project units:

1. Choose **Options > Project Options**. The Project Options dialog box displays.
2. Click the **Global Units** tab and verify that your settings match those in the following figure. You can choose units by clicking the arrows to the right of the display boxes.



3. Click **OK** to save your settings. If a warning message displays, click **OK**.

Creating a Schematic

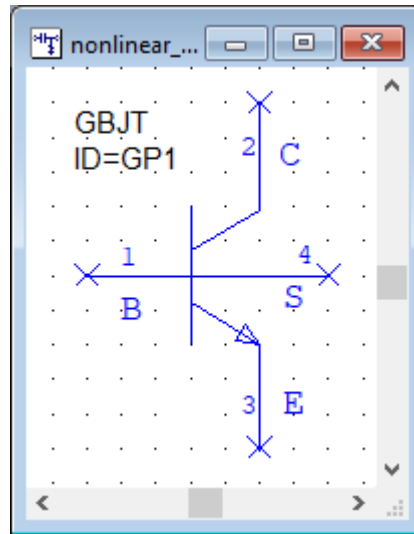
To create a schematic:

1. Choose **Project > Add Schematic > New Schematic**. The New Schematic dialog box displays.
2. Type "**IV Curve**", and click **Create**. A schematic window displays in the workspace.

Placing a Nonlinear Model from the Library

To place a nonlinear model:

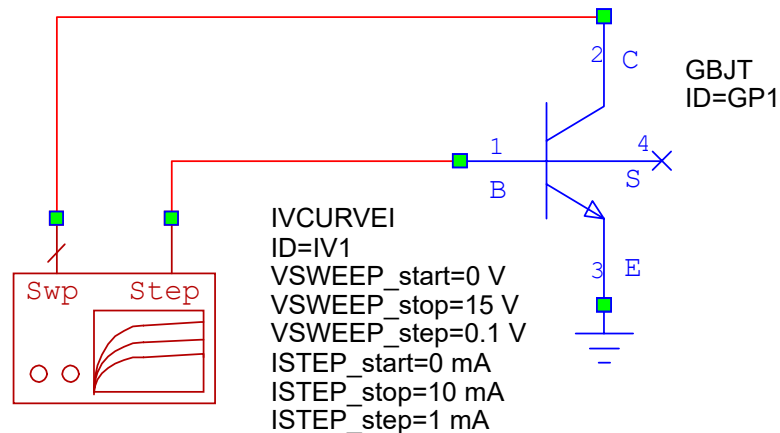
1. Click the **Elements** tab to display the Element Browser.
2. Expand the **Nonlinear** category, then click the **BJT** group. Select the GBJT model and place it on the schematic.
3. Double-click GBJT in the schematic to display the Element Options dialog box. On the **Parameters** tab, click **Show Secondary**. Note that the parameters are set to default values. You can change the parameters as per devices used in a design. Click **OK**. In this example you use a model with its parameters already changed for convenience. In the Project Browser, right-click **Circuit Schematics** and choose **Import Schematic**. The Browse for File dialog box displays.
4. Navigate to the *C:\Program Files\AWR\AWRDE\I4\Examples* or *C:\Program Files (x86)\AWR\AWRDE\I4\Examples* directory.
5. Select the *nonlinear_start.sch* file and click **Open** to import and open the schematic.



Placing an IV Curve Meter on the Nonlinear Element

To place an IV curve meter:

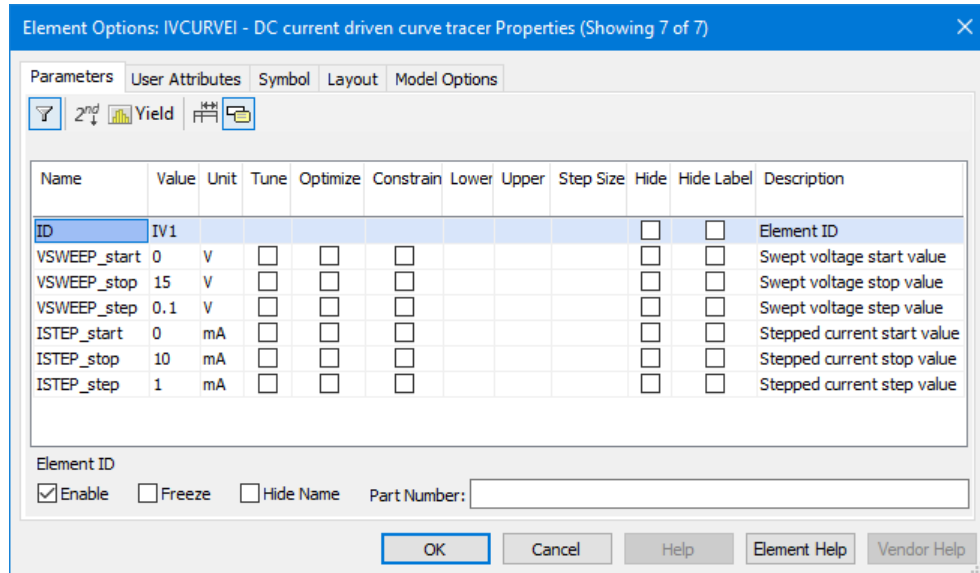
1. Click the "IV Curve" schematic window to make it active.
2. In the Element Browser, expand the **MeasDevice** category, then click the **IV** group. Select the IVCURVEI model and place it in the "IV Curve" schematic as shown in the following figure. You can select its parameter block and move it as shown.
3. Place the cursor over the Step node of IVCURVEI. The cursor displays as a wire coil symbol. Click, then drag the cursor to node 1 of the GBJT transistor, and click to place the wire.
4. Repeat step 3 to connect the Swp node of IVCURVEI to node 2 of the GBJT transistor.
5. Choose **Draw > Add Ground**, or click the **Ground** button on the toolbar. Move the cursor into the schematic window and position the ground on node 3 of the GBJT transistor, then click to place it.



Editing the IV Curve Meter Element

To specify IVCURVEI settings:

1. In the Schematic window, double-click the IVCURVEI element. The Element Options dialog box displays.
2. Edit the parameters to the values shown in the following figure, then click **OK**.

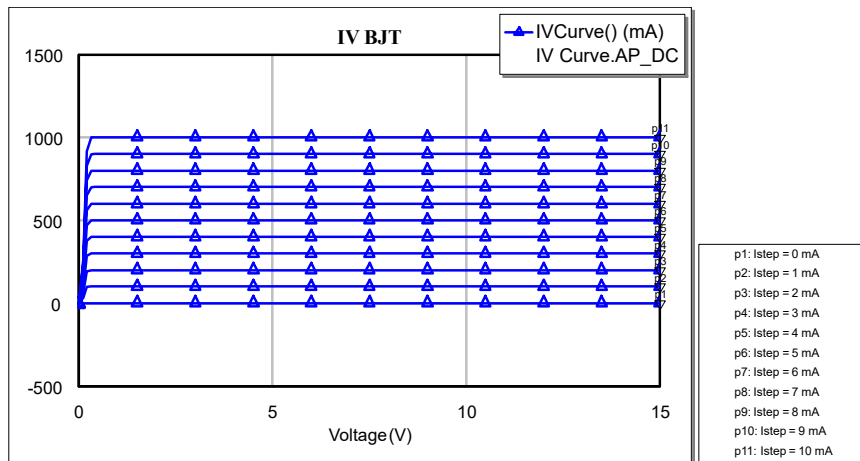


Adding an IV Curve Measurement

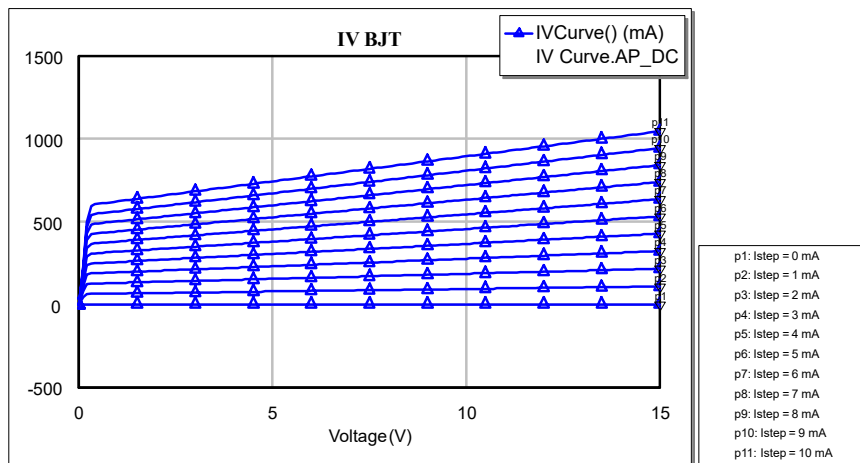
To create a graph and add an IV curve measurement:

1. In the Project Browser, right-click **Graphs** and choose **New Graph**. The New Graph dialog box displays.
2. Type "**IV BJT**" as the graph name, select **Rectangular** as the graph type, and click **Create**. The graph displays in the workspace.
3. Right-click the "IV BJT" graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays.
4. Select **Nonlinear > Current** in **Measurement Type** and **IVCurve** in **Measurement**. Select **APLAC DC** as the **Simulator**, select **IVCurve** as the **Data Source Name**, and click **OK**.
5. Choose **Simulate > Analyze**. The simulation response in the following graph displays.

NOTE: You can disable parameter markers by right-clicking the graph and choosing **Options**. On the **Markers** tab of the Rectangular Plot Options dialog box, clear the **Param markers enabled** and **Param markers in legend** check boxes.



6. Click the "IV Curve" schematic window to make it active.
7. Click the GBJT element to select it and press **Delete** to delete it.
8. Click the "nonlinear_start" schematic window to make it active.
9. Click the GBJT element to select it and press **Ctrl+C** to copy it.
10. Click the "IV Curve" schematic window to make it active.
11. Press **Ctrl+V** to copy it to the "IV Curve" schematic. Wire the circuit as shown in the previous example.
12. Choose **Simulate > Analyze**. The simulation response shown in the following graph displays. Notice the change in the IV curve graph.

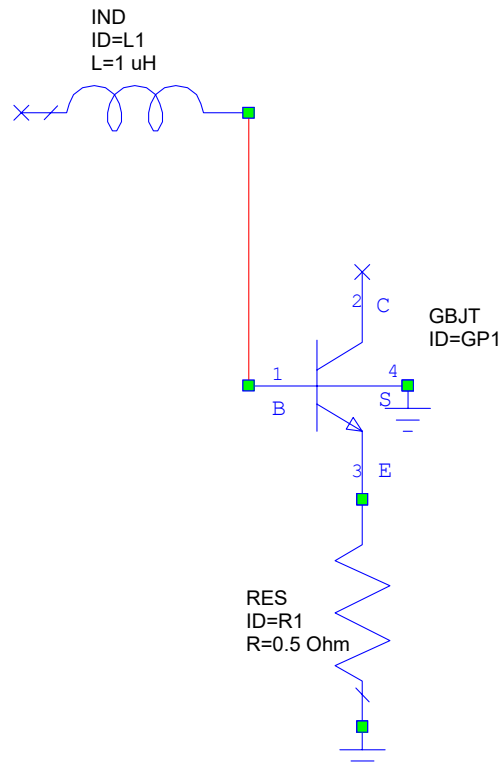


Creating a Bias Circuit

To create a DC bias circuit:

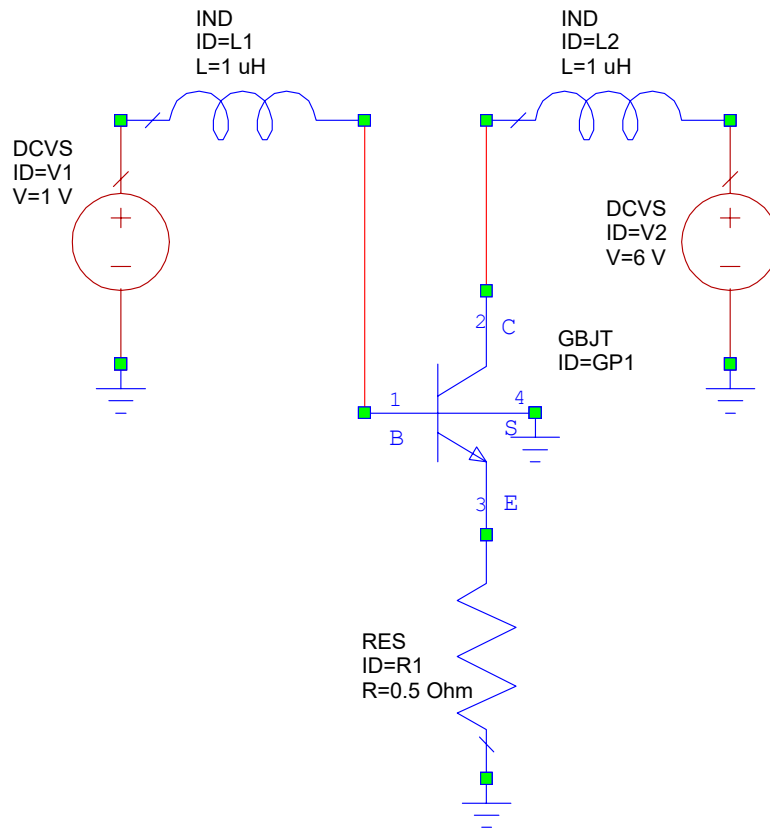
1. In the Project Browser, right-click the "nonlinear_start" schematic and choose **Rename Schematic**. In the Rename Schematic dialog box rename the schematic to "**DC Bias**", and then click **Rename**. Make DC Bias the active window.

2. In the Element Browser, expand the **Lumped Element** category, then click the **Inductor** group. Select the IND model and place it above and to the left of the GBJT transistor as shown in the following figure.
3. Place the cursor on node 1 of the GBJT transistor. The cursor displays as a wire coil symbol. Click, then move the cursor to the right node of IND, and click once more to place the wire.
4. Double-click the IND model and set the L parameter to "1", then click **OK**.
5. Under **Lumped Element**, click the **Resistor** group. Select the RES model and place it as shown in the following figure after right-clicking once to rotate the element.



6. Double-click the RES model and set the R parameter to "0.5", then click **OK**.
7. Choose **Draw > Add Ground**, or click the **Ground** button on the toolbar and position the ground on the bottom of RES R1 as shown in the previous figure.
8. Repeat step 7, positioning the ground on node 4 of the GBJT transistor as shown in the previous figure.
9. Expand the **Sources** category, then click the **DC** group. Select the DCVS model, place it and wire it as shown on the left in the figure.
10. Click the **Ground** button on the toolbar and position the ground on the negative terminal of DCVS V1 as shown in the following figure.
11. Click IND L1 in the Schematic window. Press **Ctrl+C**, then **Ctrl+V** to copy and paste the inductor as shown in the following figure.
12. Connect the new IND element to node 2 of the GBJT model as shown in the following figure.
13. Copy the DCVS model and place the copy on the open node of IND L2 as shown in the following figure.
14. Double-click the DCVS V2 model and set the V parameter to "6", then click **OK**.

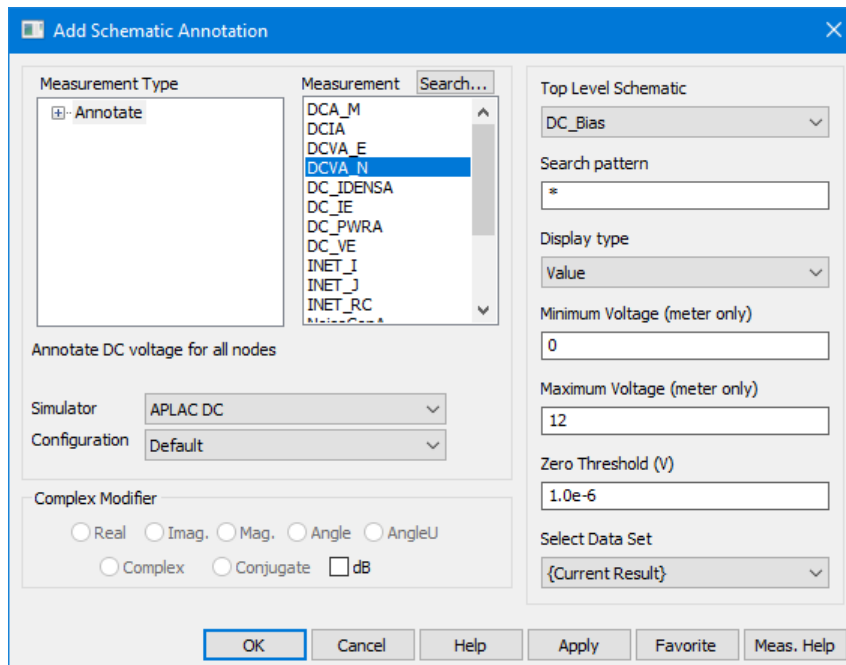
15. Click the **Ground** button on the toolbar and position the ground on the negative node of DCVS V2 as shown in the following figure.



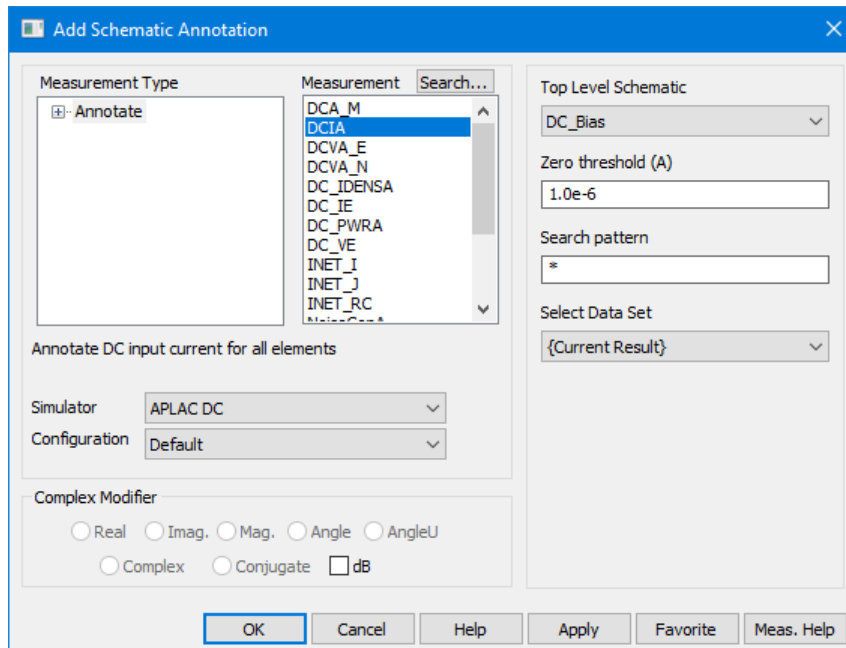
Adding Schematic Back Annotation

To add schematic back annotation to display DC voltage and current measurements:

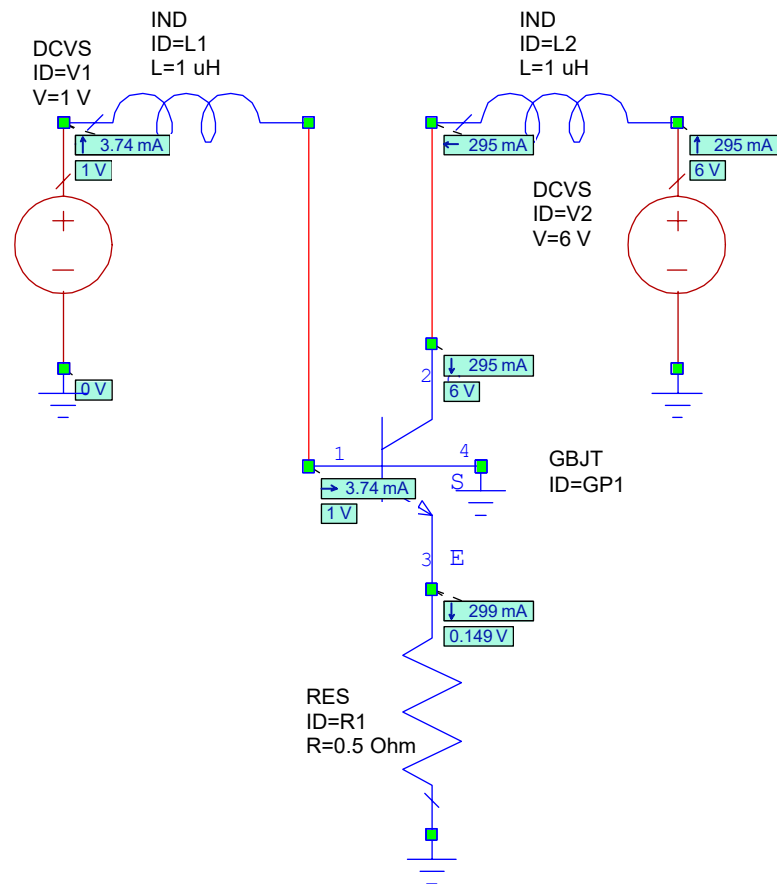
1. Right-click the "DC Bias" schematic in the Project Browser and choose **Add Annotation**. The Add Schematic Annotation dialog box displays.
2. Specify a voltage measurement using the settings in the following figure, then click **Apply**.



3. Specify a current measurement using the settings in the following figure, then click **OK**.



4. Choose **Simulate > Analyze**. The voltage displays at all nodes and the current displays at each element as shown in the following figure.



Adding a Harmonic Balance Port

Before adding a harmonic balance port, you must add DC blocking capacitors to the transistor input and output.

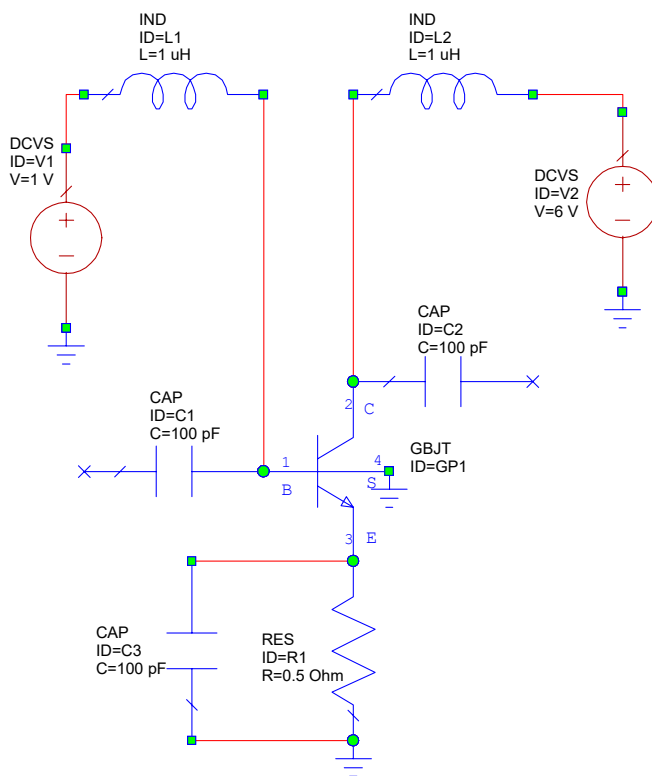
To add DC blocking capacitors:

1. Click the "DC Bias" schematic window in the workspace to make it active.
2. In the Element Browser, expand the **Lumped Element** category, then click the **Capacitor** group. Select the **CAP** model and connect it to node 1 of the GBJT transistor as shown in the following figure.
3. Double-click the CAP model and set the C parameter to "100", then click **OK**.
4. Copy the CAP model (hereinafter referred to as CAP C1) and connect the copy (CAP C2) to node 2 of the GBJT transistor as shown in the following figure.

You must also add an RF bypass capacitor across the emitter resistor.

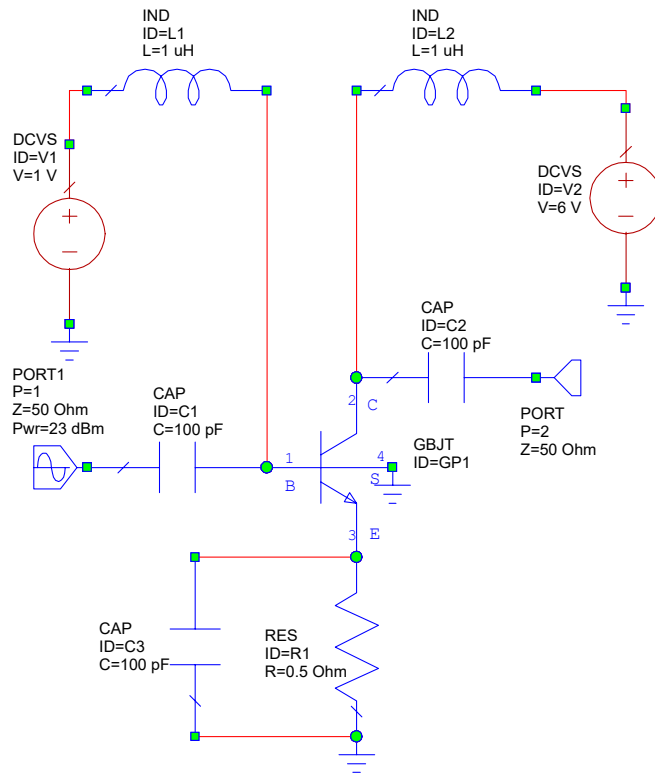
To add an RF bypass capacitor:

1. Copy the CAP C1 model and place the copy (CAP C3) to the left of RES R1 after right-clicking once to rotate the model, as shown in the following figure.
2. Connect the top node of CAP C3 to node 3 of the GBJT transistor.
3. Connect a wire between the bottom node of CAP C3 and ground.



To add a harmonic balance port:

1. In the Element Browser, expand the **Ports** category, then click the **Harmonic Balance** group. Select the PORT1 model and connect it to CAP C1 as shown in the following figure.
2. Double-click PORT1 and set the Pwr parameter to "23", then click **OK**.
3. Choose **Draw > Add Port**, or click the **Port** button on the toolbar and add a port to the open node of CAP C2 (output blocking) as shown in the following figure. This port is considered a termination port.



Specifying Nonlinear Simulation Frequencies

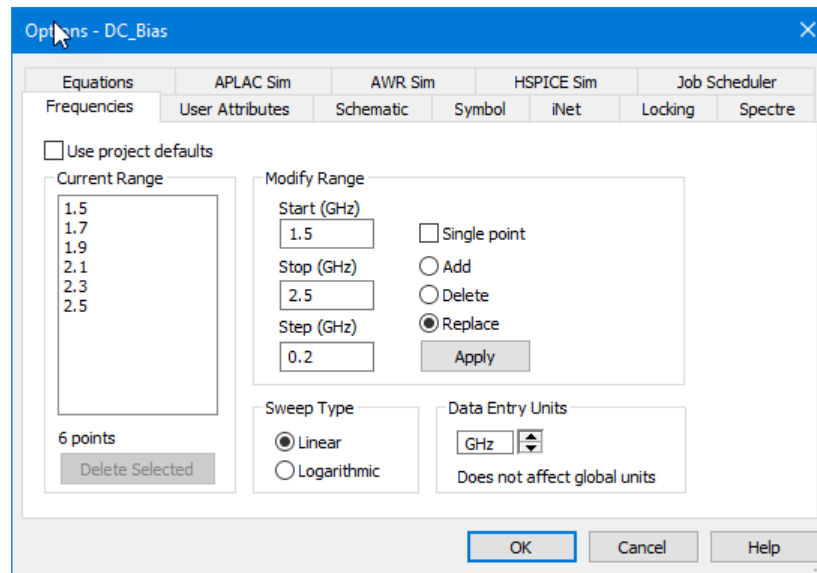
You can specify simulation frequencies in several ways:

- By defining a global sweep in the project options (choose **Options > Project Options** then click the **Frequencies** tab)
- By defining a document sweep, (right-click the schematic node in the Project Browser and choose **Options**, then click the **Frequencies** tab)
- By placing a SWPFRQ component on the schematic (located under **Circuit Elements** in the **Simulation Control** category).

You select the frequency sweep used by the simulator in the Add/Modify Measurement dialog box, as shown in [“Adding a Large Signal Reflection Coefficient Measurement”](#). The following steps define the frequency sweep using a document sweep, which is the simplest means.

To specify a document frequency sweep:

1. In the Project Browser under **Circuit Schematics**, right-click "DC Bias" and choose **Options**. The Options dialog box displays.
2. Click the **Frequencies** tab.
3. Clear the **Use project defaults** check box, select **GHz** as the **Data Entry Units**, select **Replace**, specify the **Start**, **Stop** and **Step** values shown in the following figure, click **Apply** to display the values in **Current Range**, and then click **OK**.



Adding a Large Signal Reflection Coefficient Measurement

Microwave Office software makes it easy to compute large-signal network parameters. The following exercise demonstrates how to compute the large signal reflection coefficient and display it on a Smith Chart.

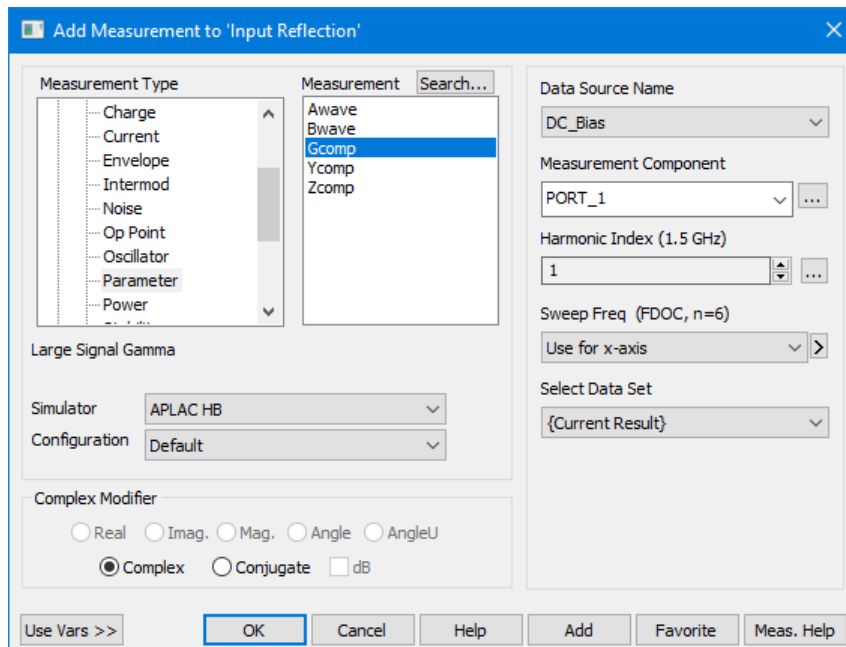
To create a Smith Chart:

1. In the Project Browser, right-click **Graphs** and choose **New Graph**. The New Graph dialog box displays.
2. Type "**Input Reflection**" as the graph name, select **Smith Chart** as the graph type, and click **Create**.

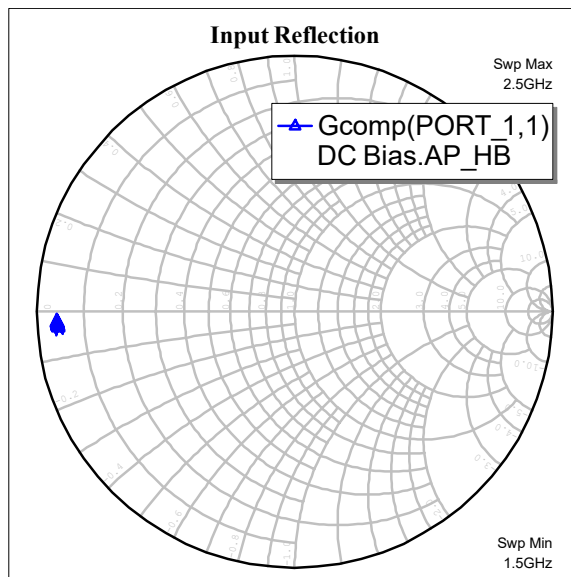
To add a large signal reflection measurement:

1. Right-click "Input Reflection" under **Graphs** and choose **Add Measurement**. The Add Measurement dialog box displays.
2. Select **Nonlinear > Parameter** as the **Meas. Type** and **Gcomp** as the **Measurement** and set the parameters as shown in the following figure, then click **Apply** and **OK**.

Note the frequency sweep control that simplifies control of the simulation frequencies and allows you to select among the available frequency sweeps. The default setting is the document sweep discussed in "[Specifying Nonlinear Simulation Frequencies](#)".



3. Choose **Simulate > Analyze**. The following simulation response displays on the Smith Chart.



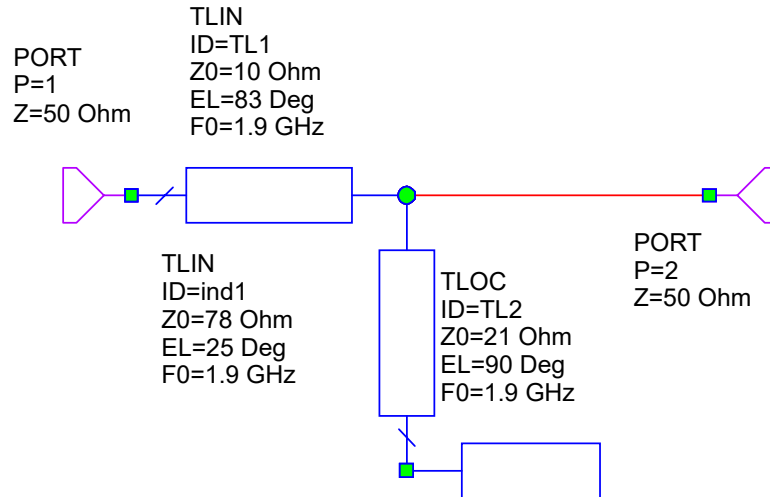
Importing Input Match and Output Match Schematics

The input and output matching for the amplifier are imported from existing schematics.

To import the input match schematic:

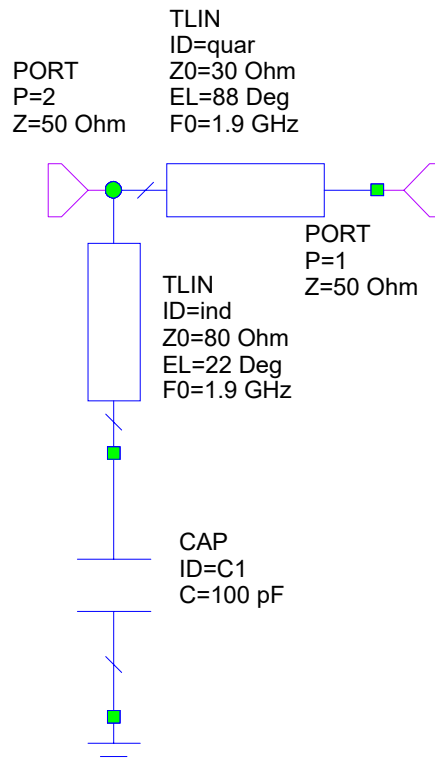
1. In the Project Browser, right-click **Circuit Schematics** and choose **Import Schematic**. The Browse For File dialog box displays.

2. Navigate to the *C:\Program Files\AWR\AWRDE\14\Examples* or *C:\Program Files (x86)\AWR\AWRDE\14\Examples* directory.
3. Select the *input_match.sch* file and click **Open** to import and open the schematic.



To import the output match schematic:

1. Right-click **Circuit Schematics** and choose **Import Schematic**.
2. Select the *output_match.sch* file from the previously opened directory and click **Open** to import and open the schematic.

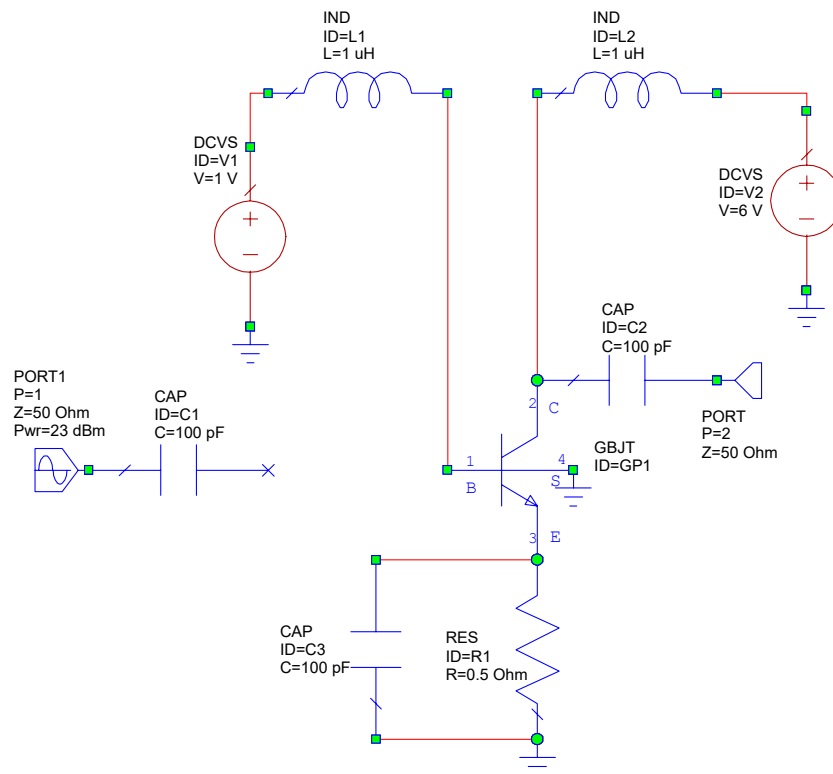


Adding Subcircuits to a Schematic

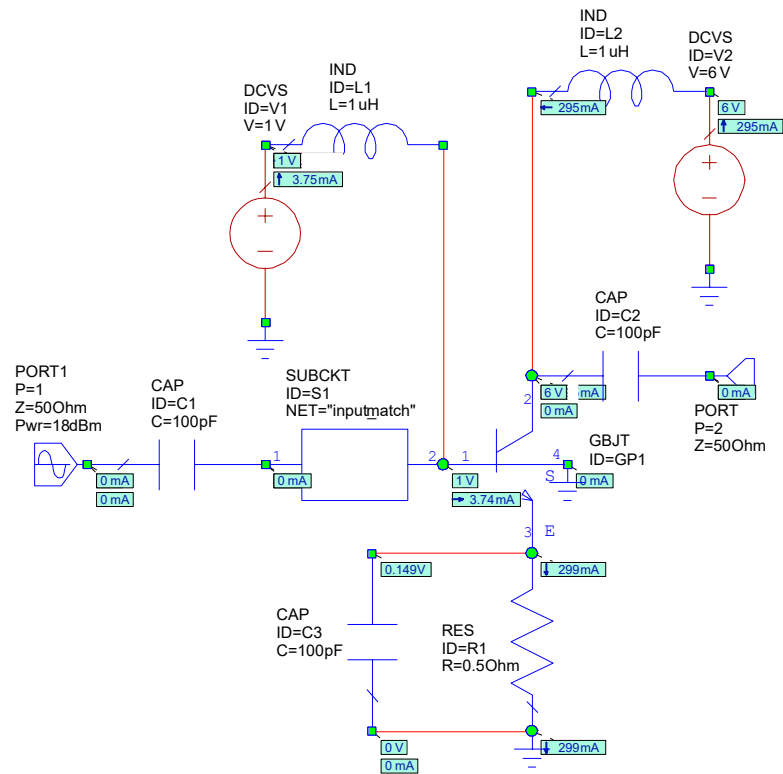
Whenever a schematic is created or imported it automatically becomes a subcircuit. You can use these subcircuits within other schematics to create circuit hierarchy.

To add the input match subcircuit to the DC Bias schematic:

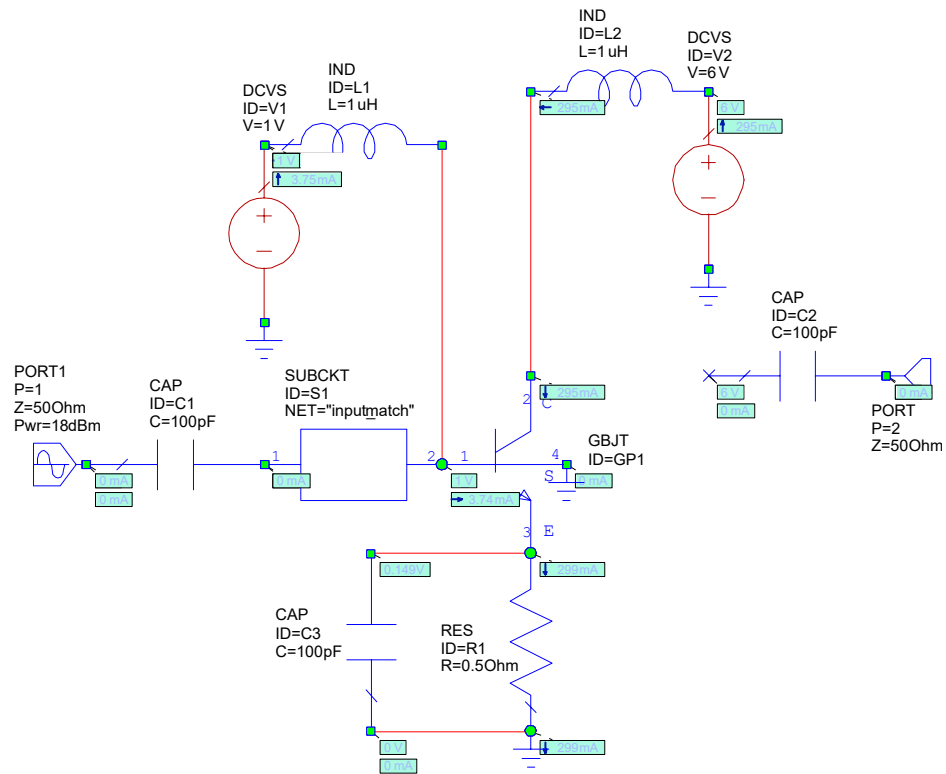
1. In the Project Browser under **Circuit Schematics**, double-click "DC Bias" to display the DC Bias schematic.
2. Click PORT1 in the schematic window, then press and hold the **Shift** key while clicking on CAP C1. Both PORT1 and CAP C1 should be selected (and are now considered one unit).
3. **Ctrl**-click on the selected elements, then drag them to the left of the circuit to break their connection with the circuit, as shown in the following figure.



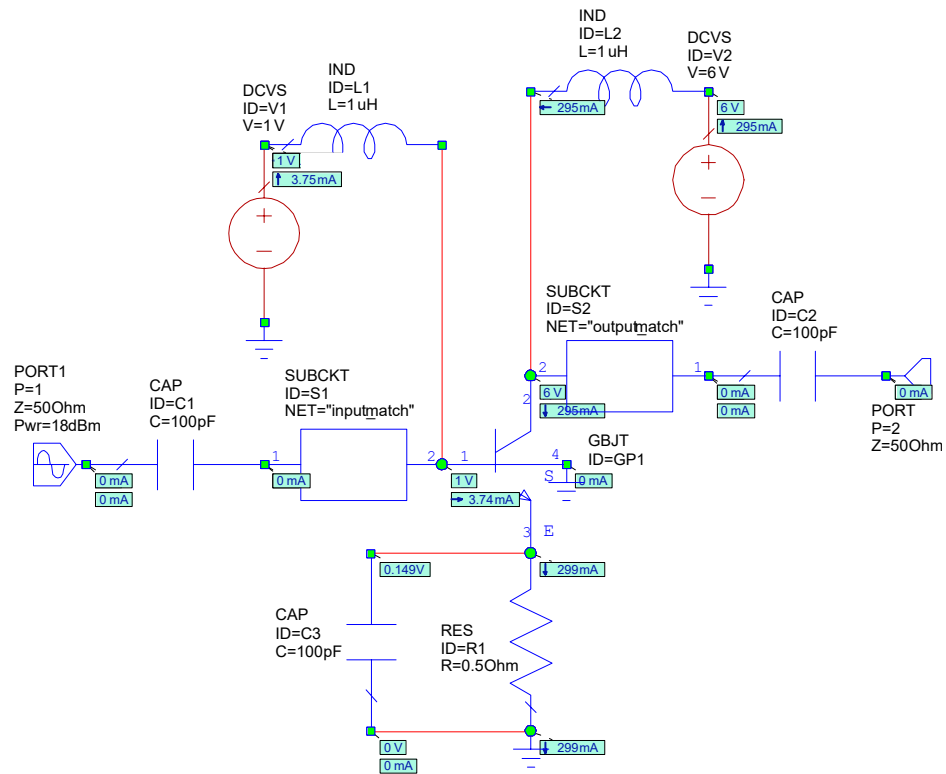
4. In the Element Browser, click the **Subcircuits** category, then select the "input match" subcircuit and place it on the schematic between CAP C1 and node 1 of the GBJT transistor as shown in the following figure.



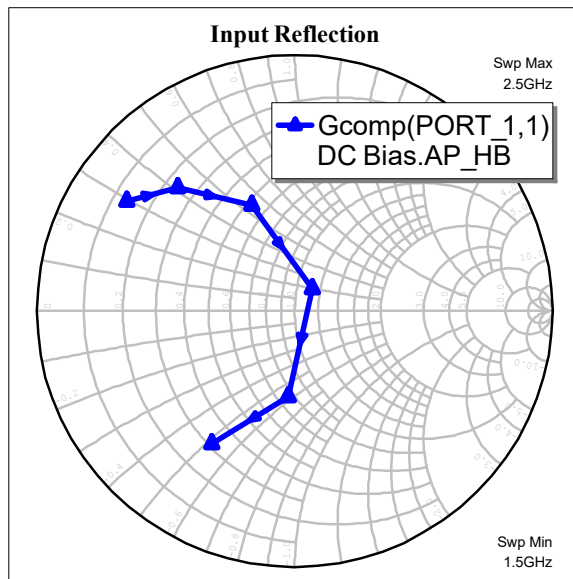
5. If the subcircuit nodes do not properly connect with the capacitor and transistor you may need to slightly move the elements until the proper connections are made.
6. Repeat steps 2 and 3 with PORT2 and CAP C2 as shown in the following figure.



7. In the **Subcircuits** category, select the "output match" subcircuit and connect it to the open node of CAP C2 *after right-clicking twice to invert it*, as shown in the following figure.
8. Connect node 2 of the output match subcircuit to node 2 of the GBJT transistor.
9. Double-click the Pwr parameter value of the PORT1 element. An edit box displays over the value. Type "**18**" to change the value from 23 to 18 dBm, then click outside the edit box to save the change.



10. Choose **Simulate > Analyze**. The simulation response in the following Smith Chart displays with the circuit matched.

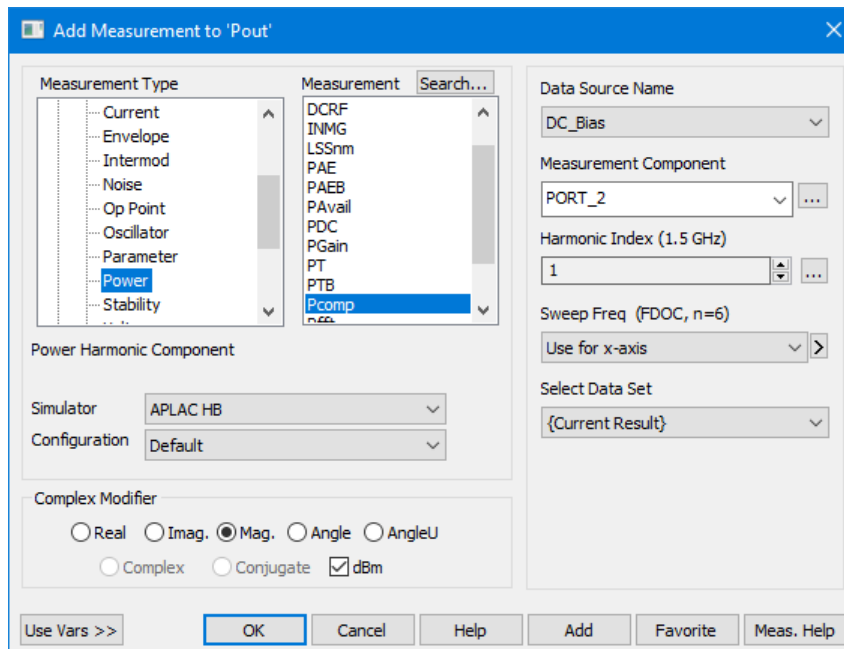


Creating a Pout vs. Frequency Measurement

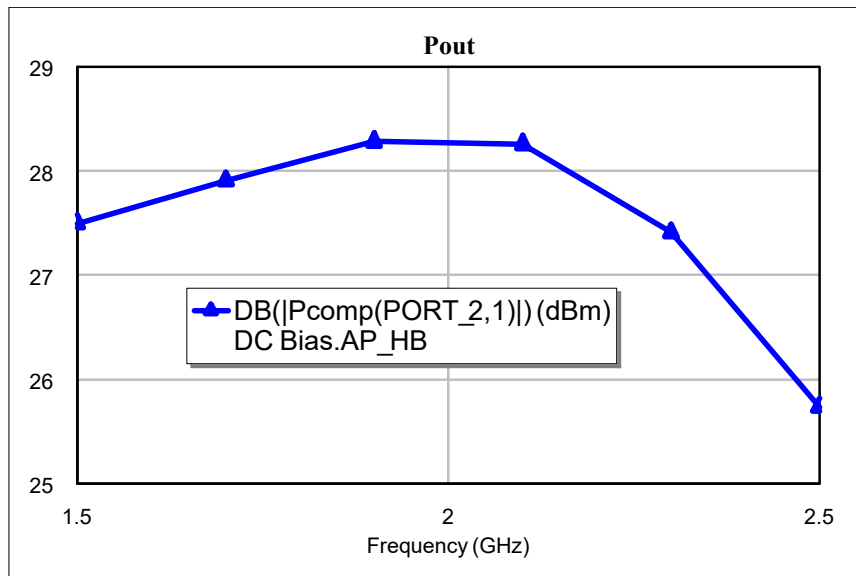
To create a graph and add a measurement (Pout versus frequency):

Creating a Power Amplifier Circuit

1. In the Project Browser, right-click **Graphs** and choose **New Graph**. The New Graph dialog box displays.
2. Type "**Pout**" as the graph name, select **Rectangular** as the graph type, and click **Create**.
3. In the Project Browser, right-click "Pout" and choose **Add Measurement**. The Add Measurement dialog box displays.
4. Create a measurement using the settings in the following figure, then click **OK**.



5. Choose **Simulate > Analyze**. The simulation response in the following graph displays.

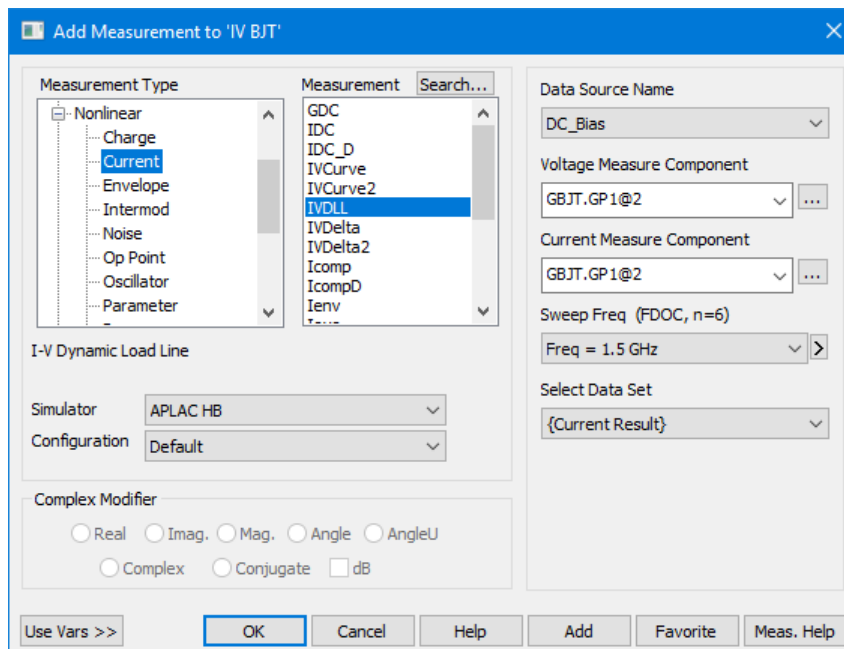


Creating a Dynamic Load Line Measurement

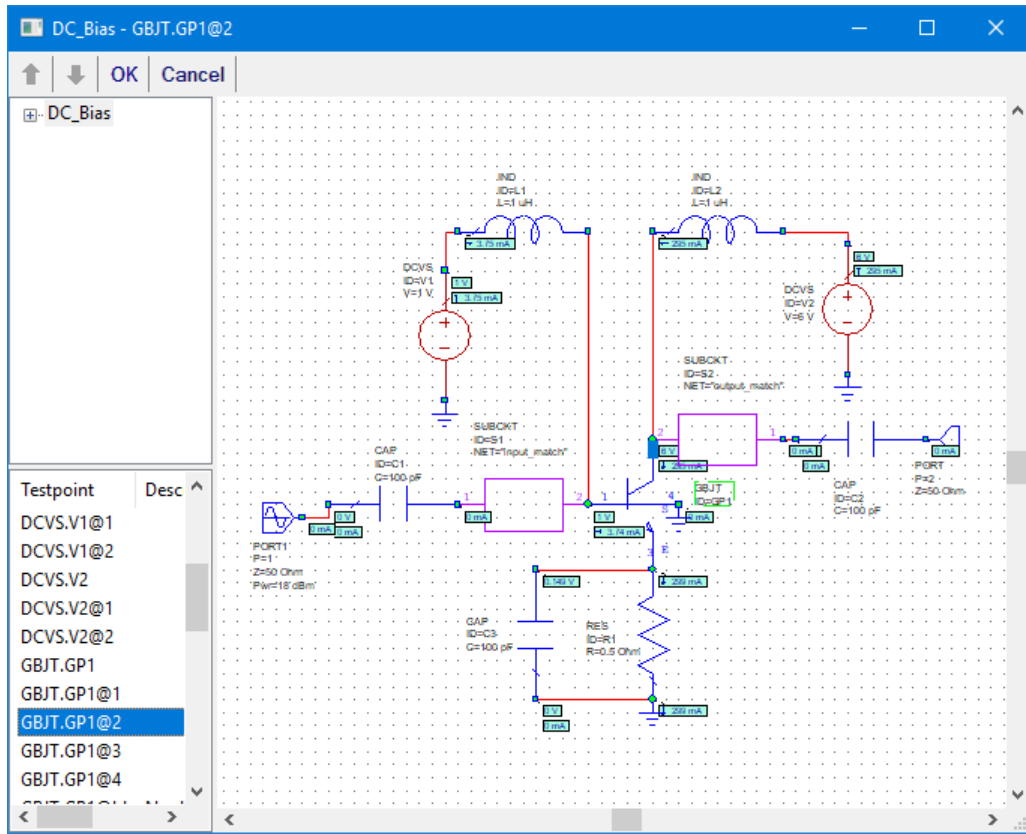
A dynamic load line measurement plots the large signal performance of the circuit superimposed on the IV curve of the device.

To create a dynamic load line measurement:

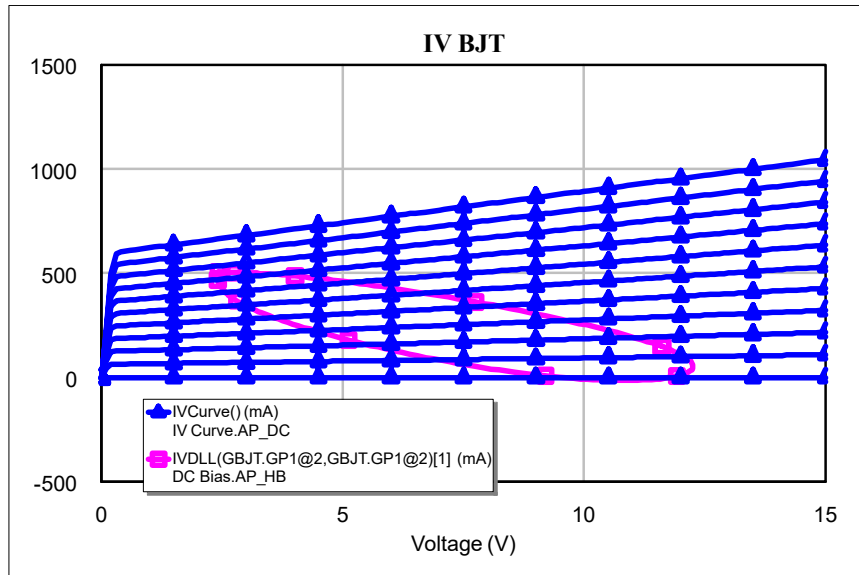
1. In the Project Browser, right-click "IV BJT" under **Graphs** and choose **Add Measurement**. The Add Measurement dialog box displays.
2. Create a measurement using the settings in the following figure, then click **OK**.



To select the **Voltage Measure Component** and **Current Measure Component**, click the "..." button to the right of these options. The Component Browser shown in the following figure displays to allow you to select the desired components.



3. Choose **Simulate > Analyze**. The simulation response in the following graph displays.



Setting up a Two-Tone Simulation

The following exercise involves a two-tone simulation that is swept over power at one frequency point. It requires a new schematic of the same circuit with a different port configuration. To create the new schematic, you can duplicate the existing schematic and edit the port configurations.

Copying a Schematic in the Project Browse

To duplicate the DC Bias schematic:

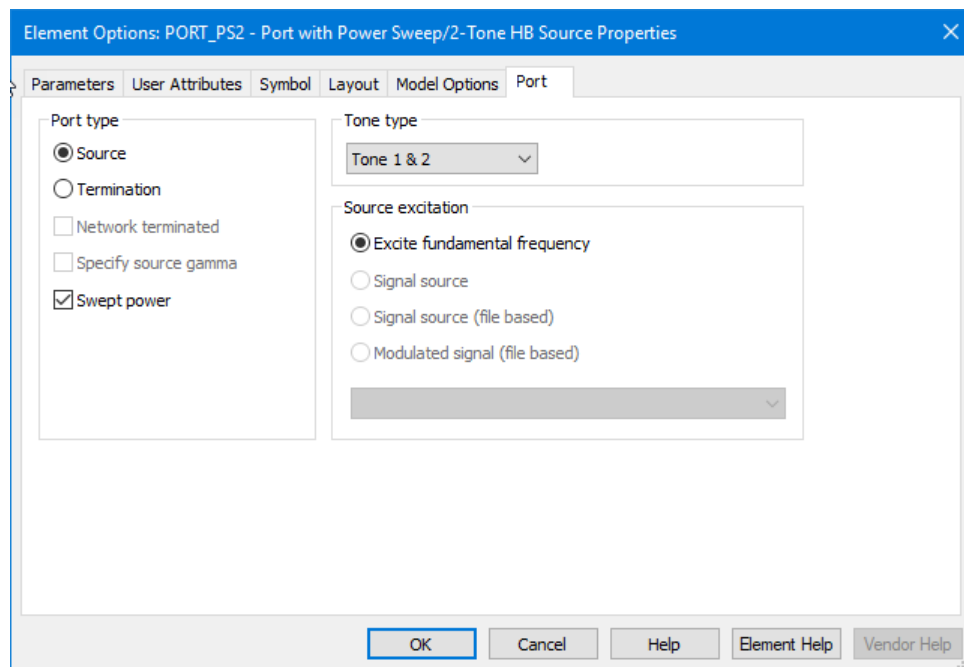
1. Click "DC Bias" under **Circuit Schematics** in the Project Browser, drag it up to the **Circuit Schematics** node and release the mouse button. A duplicate schematic named "DC Bias_1" is created.
2. Right-click the "DC Bias_1" schematic and choose **Rename**. Rename the schematic to "**Two Tone Amp**" in the Rename Schematic dialog box, and then click **Rename**.

Adding a Two-Tone Harmonic Balance Port

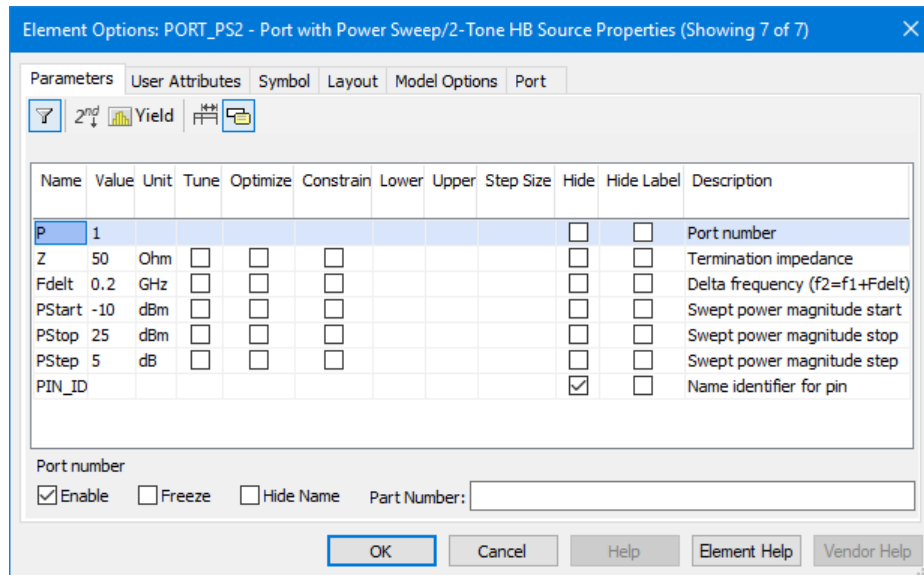
A common measurement used to characterize power amplifiers is a third-order intermodulation product versus swept power. To make this measurement, two closely-spaced tones must be injected into the input port.

To add a two-tone harmonic balance port:

1. Click the "Two Tone Amp" schematic window in the workspace to make it active. You can press the **Home** key to view the entire schematic.
2. Double-click PORT1 in the schematic window. The Element Options dialog box displays.
3. Click the **Port** tab.
4. Specify the port settings shown in the following figure.



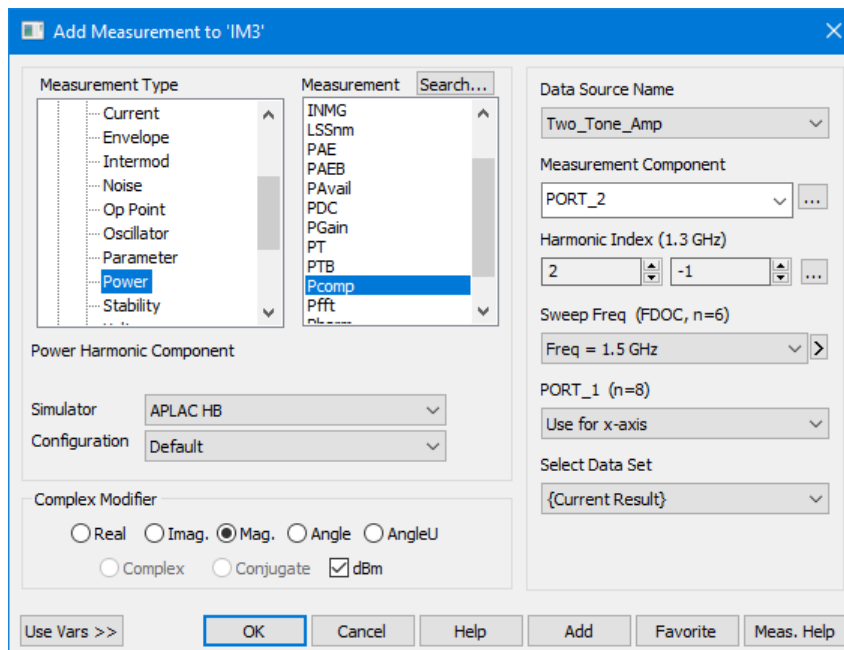
- Click the **Parameters** tab and edit the parameters to the values shown in the following figure, then click **OK**.



Adding a Third-Order Intermodulation Measurement

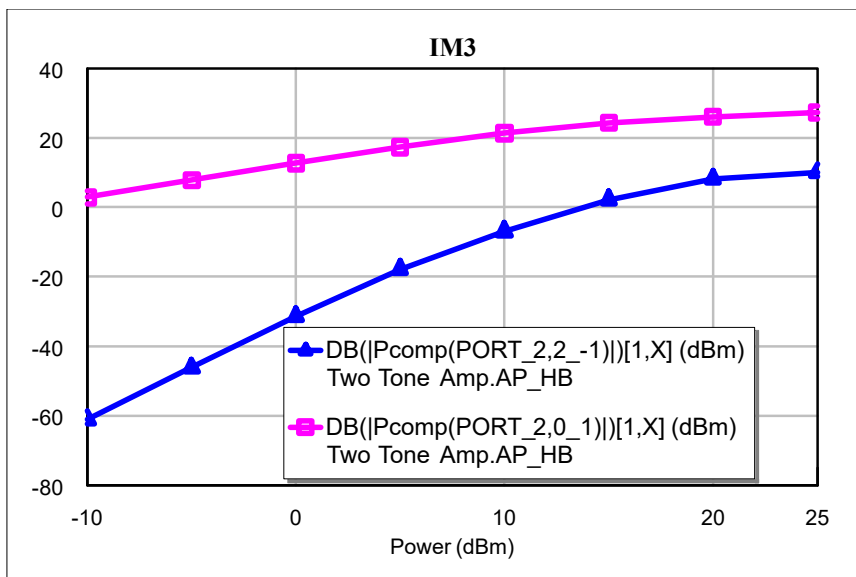
To add a third-order intermodulation product measurement:

- Create a rectangular graph named "IM3".
- Right-click "IM3" under **Graphs** and create a measurement using the settings in the following figure, then click **OK**.



- Repeat step 2 to plot a fundamental component output power, this time selecting "0" and "1" under **Harmonic Index**.

4. Choose **Simulate > Analyze**. The simulation response in the following graph should display.



Using Variable Sweeps to Measure IP3 vs Voltage

The output-referred IP3 measurement extrapolates the low-power results, so in this exercise you create another schematic to avoid multi-dimensional (voltage and power) sweeps.

To create a schematic for the IP3 measurement:

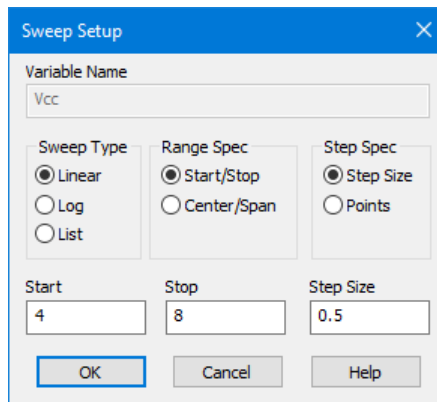
1. Copy the "Two Tone Amp" schematic under **Circuit Schematics**.
2. Right-click the "Two Tone Amp_1" schematic and rename it "**IP3**".

To change the port excitation to a fixed power:

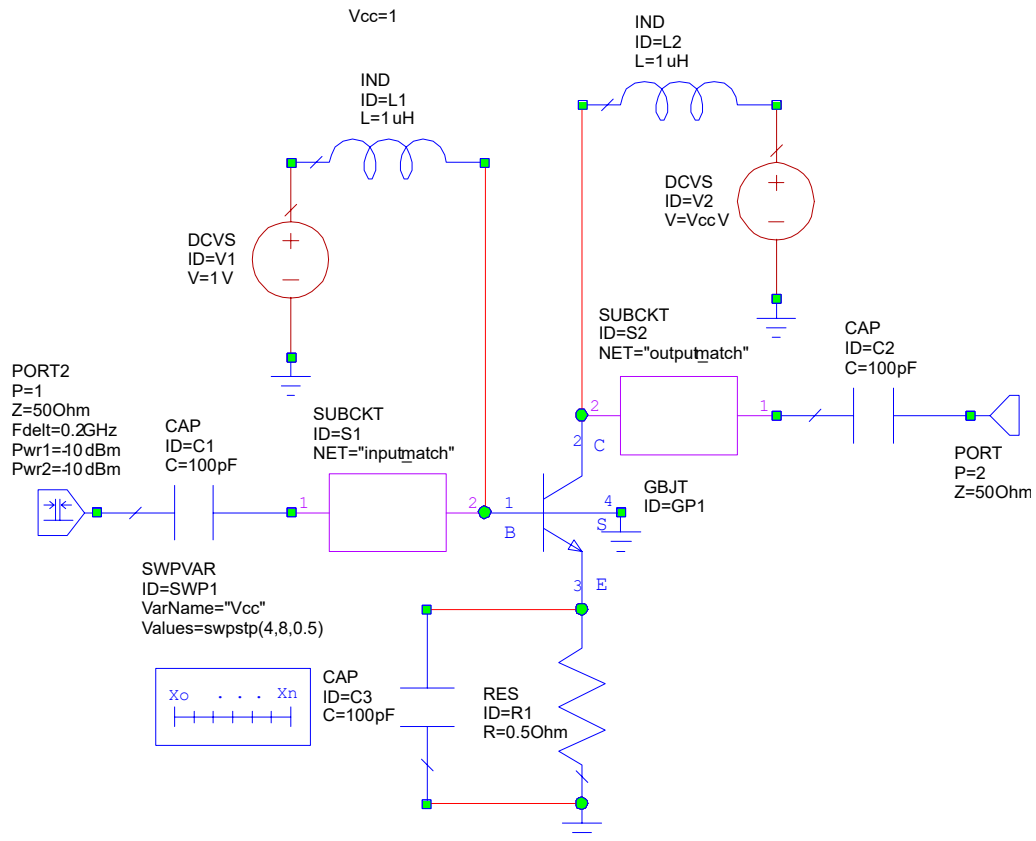
1. In the "IP3" schematic, double-click PORT_PS2. In the Element Options dialog box, click the **Port** tab and clear the **Swept power** check box, then click **OK**. The port name changes to PORT2 P=1.
2. Double-click the Pwr1 and Pwr2 parameters of this port (PORT2 P=1) and change both to "**-10**".

To define a swept variable:

1. Choose **Draw > Add Equation** or click the **Equation** button on the toolbar.
2. Move the cursor into the schematic to display an edit box. Position the box near the top of the schematic window and click to place it.
3. Type "**Vcc=1**" in the edit box, and then click outside of the box. A variable named "Vcc" is created.
4. Right-click the "Vcc" variable and choose **Setup Sweep**.
5. In the Sweep Setup dialog box, type "**4**", "**8**", and "**0.5**" for the **Start**, **Stop**, and **Step** values respectively, as shown in the following figure.



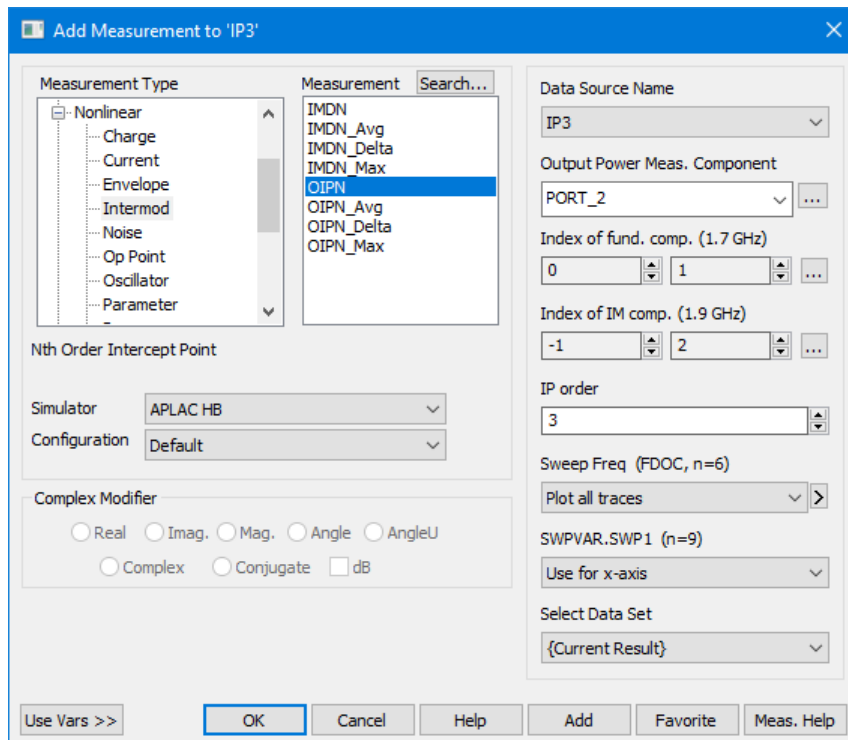
6. Click **OK** and place the SWPVAR block as shown in the following figure.



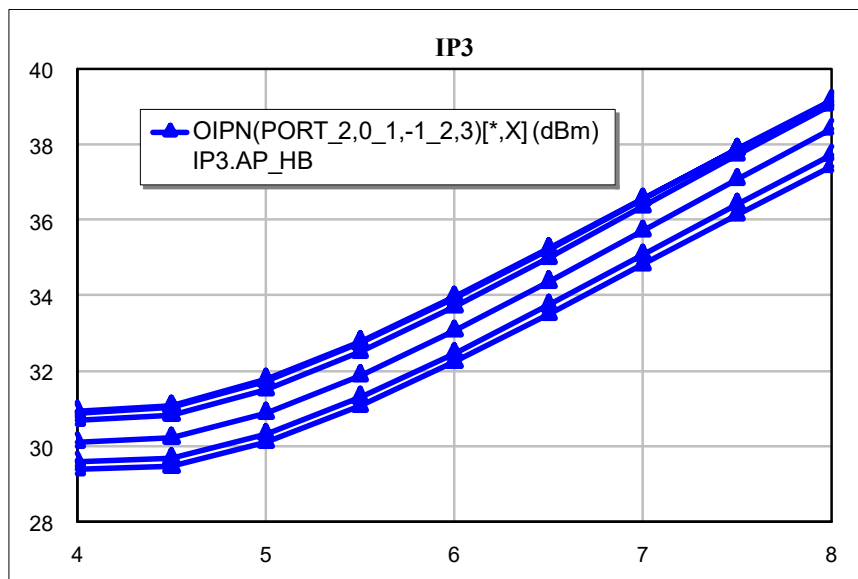
7. Double-click the V parameter value of DCVS V2 and change it to "**Vcc**".

To add a graph and measurement:

1. Add a rectangular graph named "**IP3**".
2. Create a measurement using the settings in the following figure, then click **OK**.



3. Choose **Simulate > Analyze** to obtain the output-referred IP3 intercept, swept over the bias voltage, as shown in the following graph.

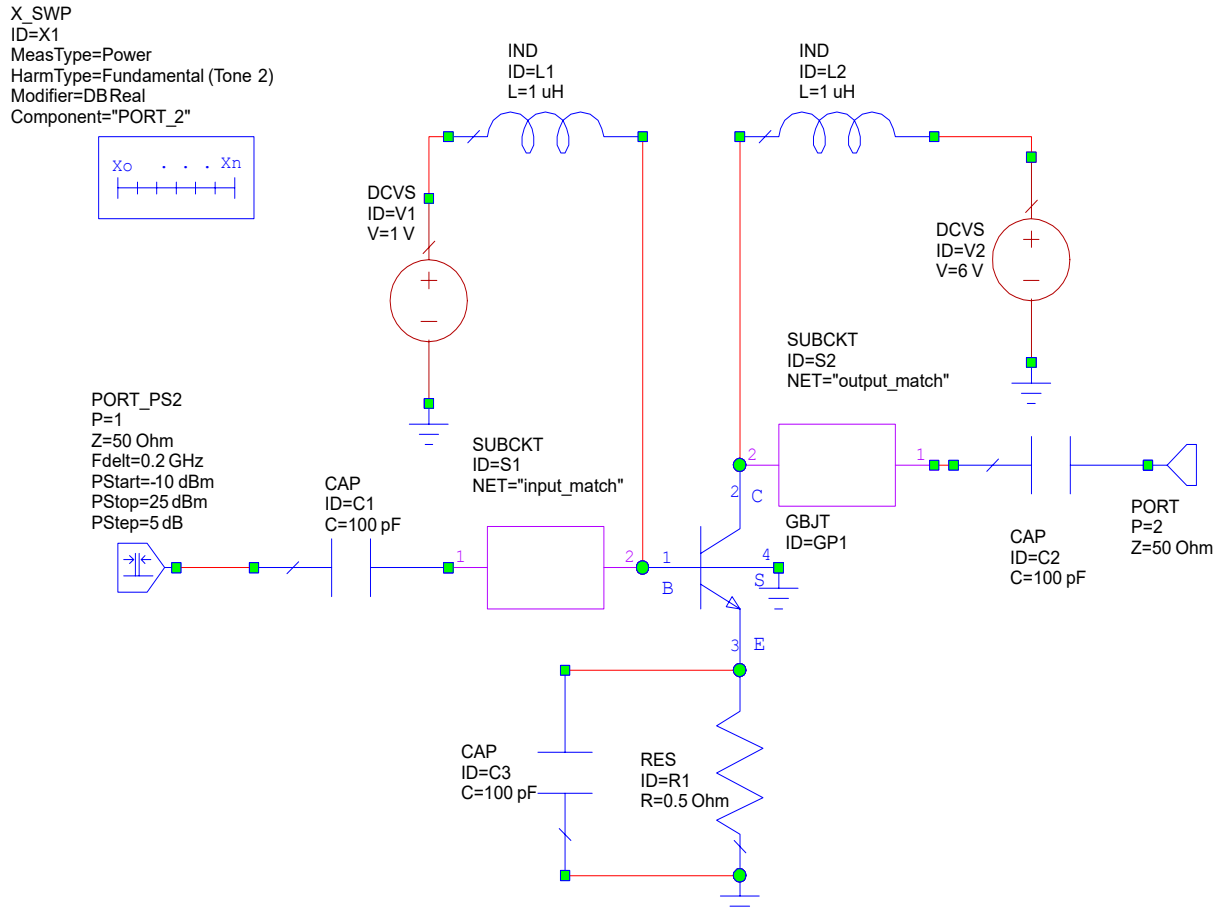


Plotting IM3 vs Output Power

To plot the IP3 vs Output Power:

Creating a Power Amplifier Circuit

1. In the Project Browser, double-click the "Two_Tone_Amp" schematic.
2. In the Element Browser, expand the **Simulation Control** category. Select the X_SWP model and place it anywhere in the schematic.

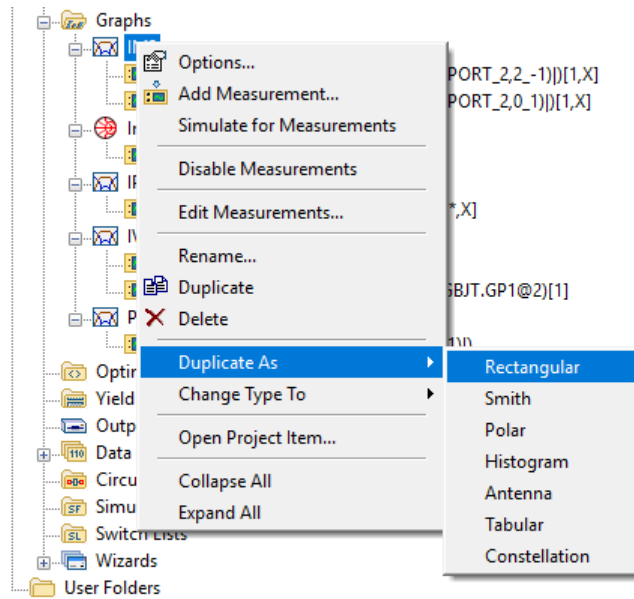


3. Set the X_SWP block parameters as shown in the following figure.

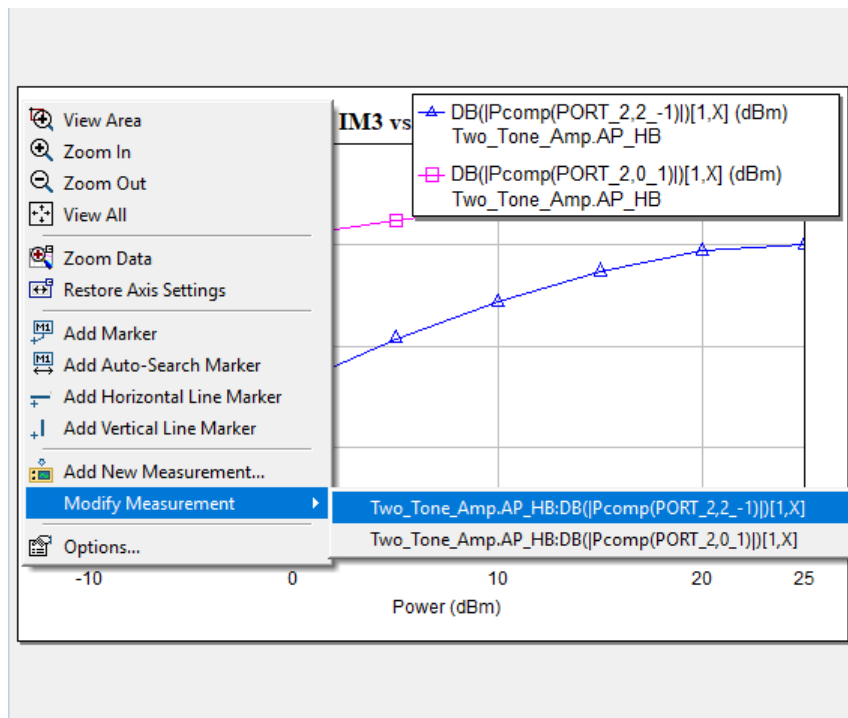
X_SWP
 ID=X1
 MeasType=Power
 HarmType=Fundamental (Tone 2)
 Modifier=DB Real
 Component="PORT_2"



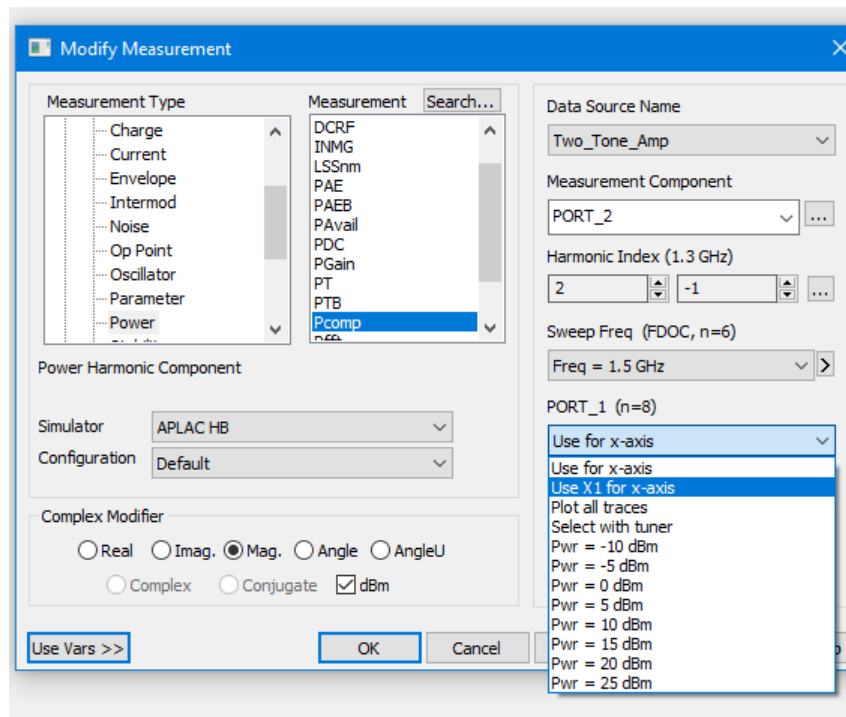
4. In the Project Browser under the **Graphs** node, right-click "IM3" and choose **Duplicate As > Rectangular**. A duplicate graph named "IM3_1" is created.



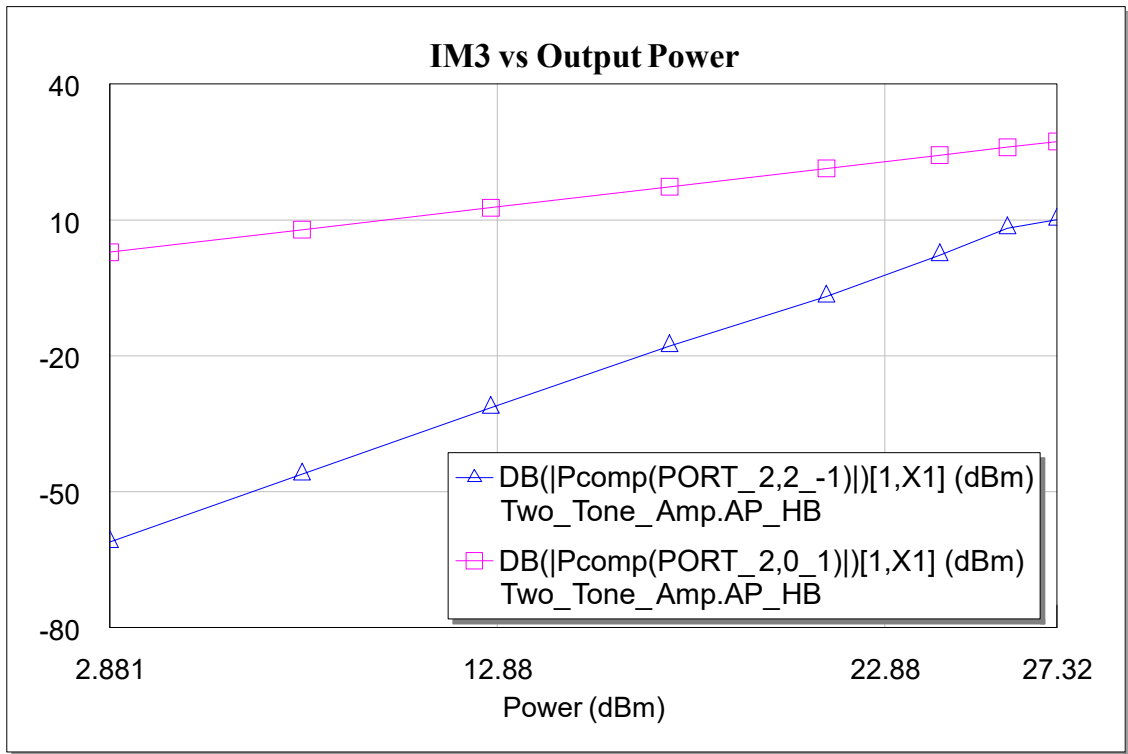
5. In the Project Browser under the **Graphs** node, right-click "IM3_1" and choose **Rename**. In the Rename Graph dialog box, rename the graph to "**IM3 vs Output Power**", and then click the **Rename** button.
6. On the "IM3 vs Output Power" graph, right-click and choose **Modify Measurement** and the first measurement in the list.



7. Modify the measurement as shown in the following figure, setting **PORT_1** to **Use X1** for x-axis.



8. Repeat step 7 for the other measurement in the "IM3 vs Output Power" graph, setting **PORT_1** to **Use X1 for x-axis**.
9. Choose **Simulate > Analyze**. The "IM3 vs Output Power" graph displays similar to the following figure. Compare the x-axis values against the y-axis values in the "IM3" graph.



10. Save and close the project.

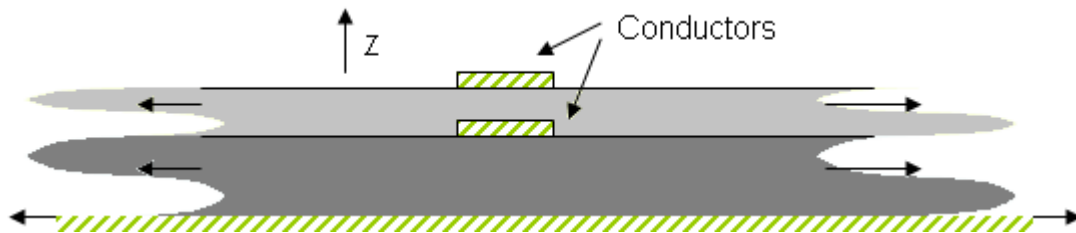
Chapter 7. Microwave Office: Using the Electromagnetic Simulator

Electromagnetic (EM) simulators use Maxwell's equations to compute the response of a structure from its physical geometry. EM simulations are ideal because they can simulate highly arbitrary structures and still provide very accurate results. In addition, EM simulators are not subject to many of the constraints of circuit models because they use fundamental equations to compute the response. One limitation of EM simulators is that simulation time grows exponentially with the size of the problem, thus it is important to minimize problem complexity to achieve timely results.

EM Simulation in Microwave Office

EM simulation and circuit simulation are complementary techniques for circuit design, and you can use the two approaches in combination to solve many design problems. Microwave Office software supports the seamless integration of many EM simulators via an "EM Socket" software interface. To integrate a third-party simulator with the NI AWR Design Environment suite, the simulator must support an interface complementary to the "EM Socket" interface. Since it is not possible to document the unique features of all third-party simulators that integrate with Microwave Office software, the example presented here is centered around NI AWR's native EM simulator, AXIEM. Manipulation of the geometry and simulation settings in the program are very similar for all simulators. Detailed documentation of the unique functionality supplied by other compatible solvers is available from the individual vendors.

AXIEM is a Method of Moments solver that solves for the currents on conductors that can be embedded in a stackup of planar dielectric layers. The dielectric layers are of infinite extent in the x-y plane as shown in the following figure. The dielectric layers are sandwiched between an infinite half-space above and infinite half-space below. The half-space below the dielectric is typically a conductor or perfect electric conductor (PEC), but it can also be an infinite open boundary if needed. The half space above the dielectric layers is typically an infinite open boundary which correctly models free-space radiation, but it can also be a conducting plane.



All conductor shapes in AXIEM must be flat shapes drawn in the x-y plane. Each shape can be extruded orthogonally in the z direction to give it a finite thickness. There are no restrictions on the thickness of the conductors; they can be infinitely thin or can have finite thickness. The thickness is allowed to protrude into one or more dielectric layers as well. There are also no restrictions on the shapes relative to the grid. A typical model built in AXIEM contains conductors and vias. Conductors can be of any thickness, and their bottom surface must rest on the top surface of a dielectric layer and can extend upward (in the positive z-direction) or downward (in the negative z-direction). Vias however, always protrude downward from the layer on which they are drawn, and they always extend through to the bottom surface of one or more complete dielectric layers. Since there are no restrictions on the height of conductors, you can draw vias using thick conductors. You should use actual vias when possible, however, because there are some options in the mesher that treat a via differently from a thick conductor that spans the same z extent.

AXIEM uses a mesh defined on the surface of the conductors as the basis for the solution. The effects of the dielectric layers are modeled using Green's functions, which provides a solution from a much smaller set of unknowns. The surface currents modeled by AXIEM include all x, y and z components. There are no restrictions on how current flows on the surfaces of conductors created in AXIEM. The ability to model all surface currents accurately allows accurate analysis of conductor traces of any thickness (even lines with greater thickness than width).

Creating a Distributed Interdigital Filter

This example demonstrates how to use Microwave Office software to simulate a distributed microstrip interdigital filter using the EM simulator.

The basic procedures in this example include:

- Creating an EM structure
- Setting up an enclosure
- Creating a layout
- Modeling via holes
- Viewing a structure in 3D
- Defining ports and de-embedding lines
- Configuring structure mesh
- Viewing current density
- Performing Advanced Frequency Sweep (AFS)
- Adding an EM structure into a schematic and simulating

NOTE: The *Quick Reference* document lists keyboard shortcuts, mouse operations, and tips and tricks to optimize your use of the NI AWR Design Environment suite. Choose **Help > Quick Reference** to access this document.

Creating a New Project

The example you create in this chapter is available in its complete form as *EM_example.emp*. To access this file from a list of Getting Started example projects, choose **File > Open Example** to display the Open Example Project dialog box, then **Ctrl-click** the **Keywords** column header and type "**getting_started**" in the text box at the bottom of the dialog box. You can use this example file as a reference.

To create a project:

1. Choose **File > New Project**.
2. Choose **File > Save Project As**. The Save As dialog box displays.
3. Navigate to the directory in which you want to save the project, type "**EM_example**" as the project name, and then click **Save**.

Importing a Layer Process File (LPF)

To import a Layer Process File (LPF):

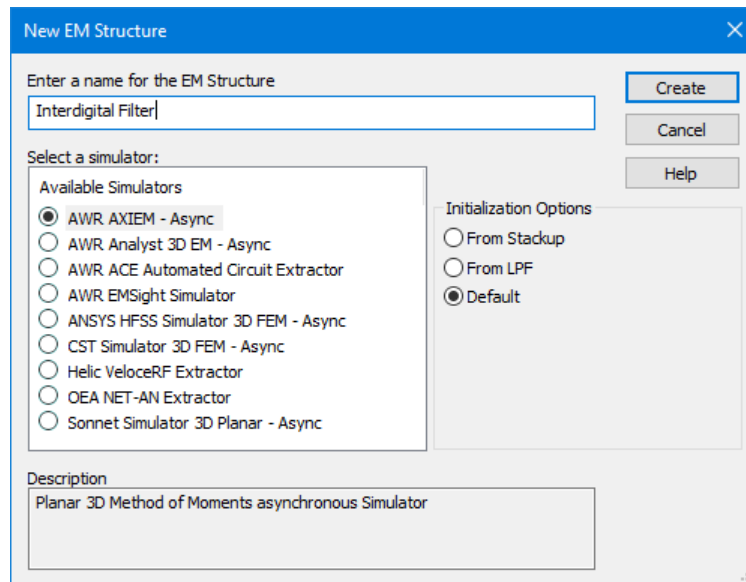
1. Choose **Project > Process Library > Import LPF**. The Import Process Definition dialog box displays.
2. Navigate to the program directory (*C:\Program Files\AWR\AWRDE\14* or *C:\Program Files (x86)\AWR\AWRDE\14* is the default installation directory). If you changed the default installation, then browse to that directory instead.
3. Choose the *Blank.lpf* file and click **Open**, then click **Replace** when prompted to replace the existing LPF.

This step is not required for EM simulation, but ensures that your project is set up with the same project units, grid spacing, database resolution, and drawing layers as this example. These settings should be set appropriately whenever you start a new design. Changing the settings after starting the design may affect the design.

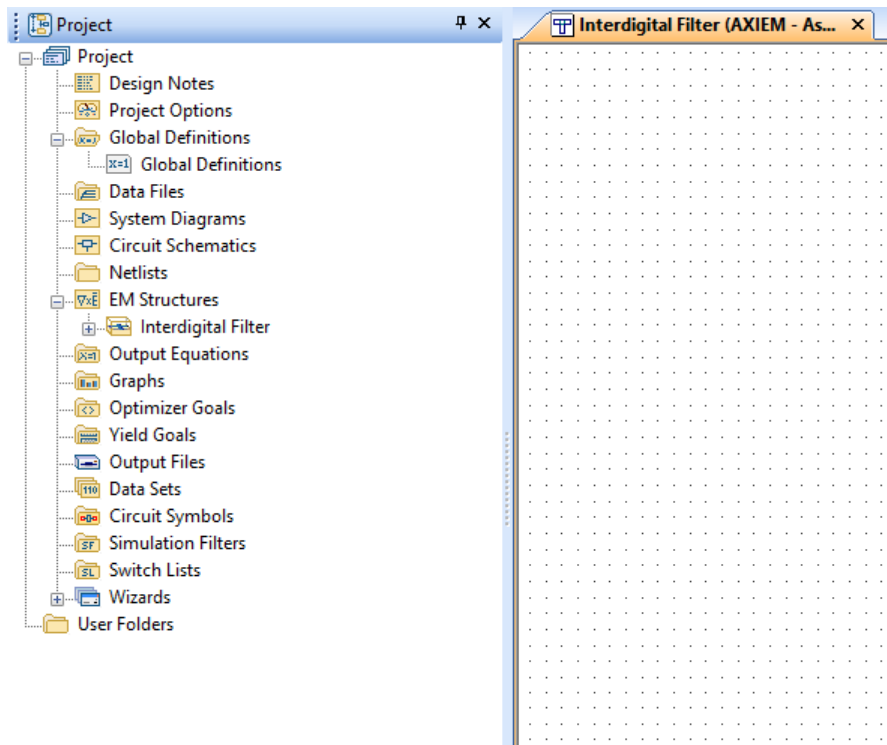
Creating an EM Structure

To create an EM structure:

1. Choose **Project > Add EM Structure > New EM Structure**. The New EM Structure dialog box displays.
2. Type "**Interdigital Filter**" and select **AWR AXIEM - Async** from the list of EM simulators available on your computer, then click **Create**.



3. An EM structure window displays in the workspace.

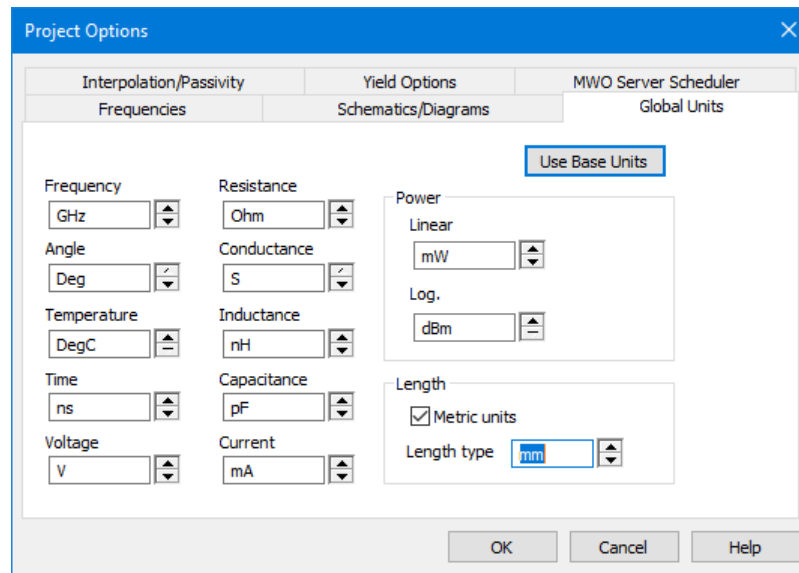


Setting Up the Enclosure

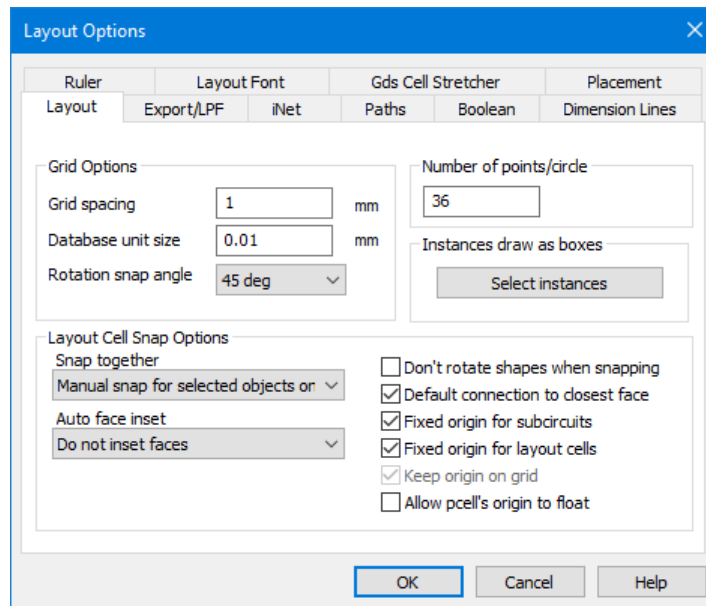
The enclosure defines the material types, the dielectric materials for each of the layers in an EM structure, the minimum grid units used to specify conductor materials in the structure, and sets the boundary conditions.

To set up the enclosure:

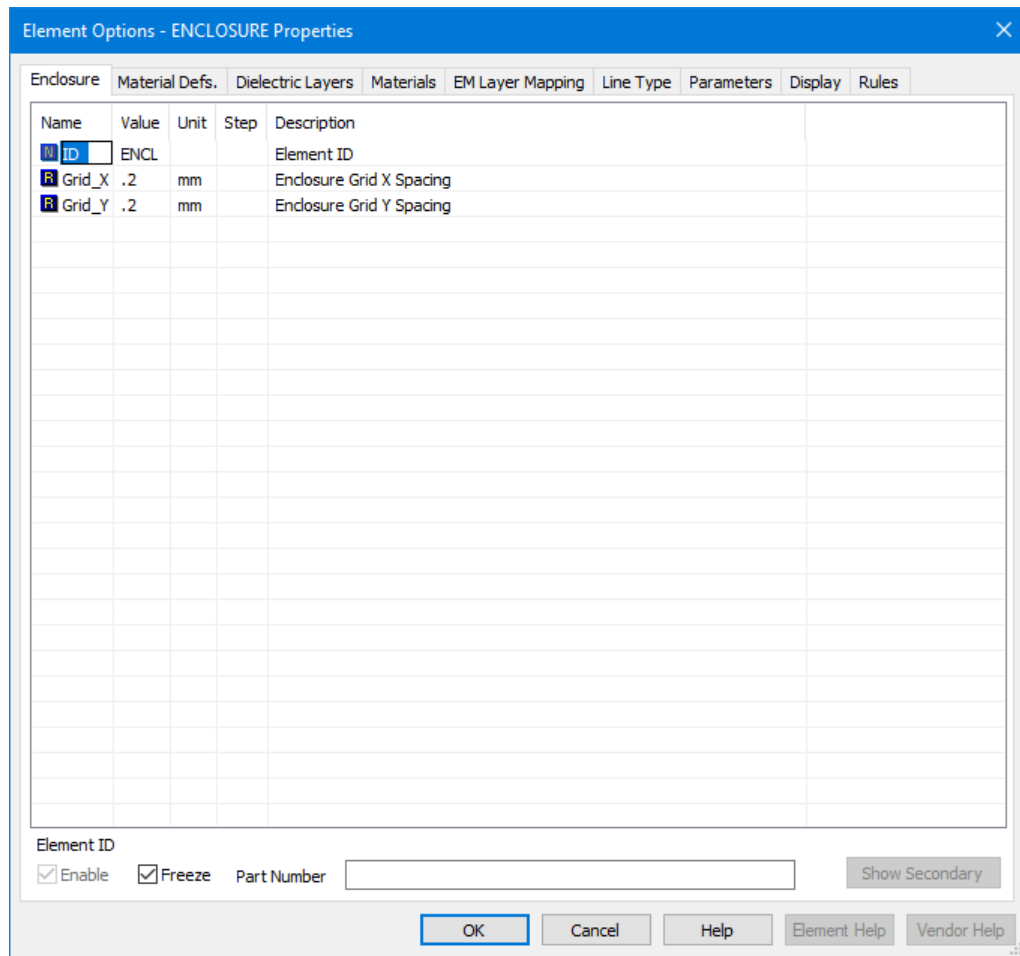
1. Choose **Options > Project Options**. The Project Options dialog box displays.
2. Click the **Global Units** tab and select the **Metric units** check box, then set the **Length type** to **mm** and click **OK**.



3. Choose **Options > Layout Options**. The Layout Options dialog box displays.
4. Click the **Layout** tab, and under **Grid Options** set the **Grid Spacing** to "1" mm and the **Database unit size** to "0.01" mm, clear any selected options, and then click **OK**. If a warning message displays, click **Yes**.

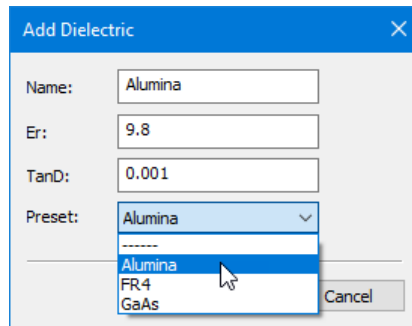


5. In the Project Browser, under **EM Structures** and "Interdigital Filter", double-click **Enclosure**. The Element Options - ENCLOSURE Properties dialog box displays.
6. Click the **Enclosure** tab and type "0.2" as the **Grid_X** value and "0.2" as the **Grid_Y** value.

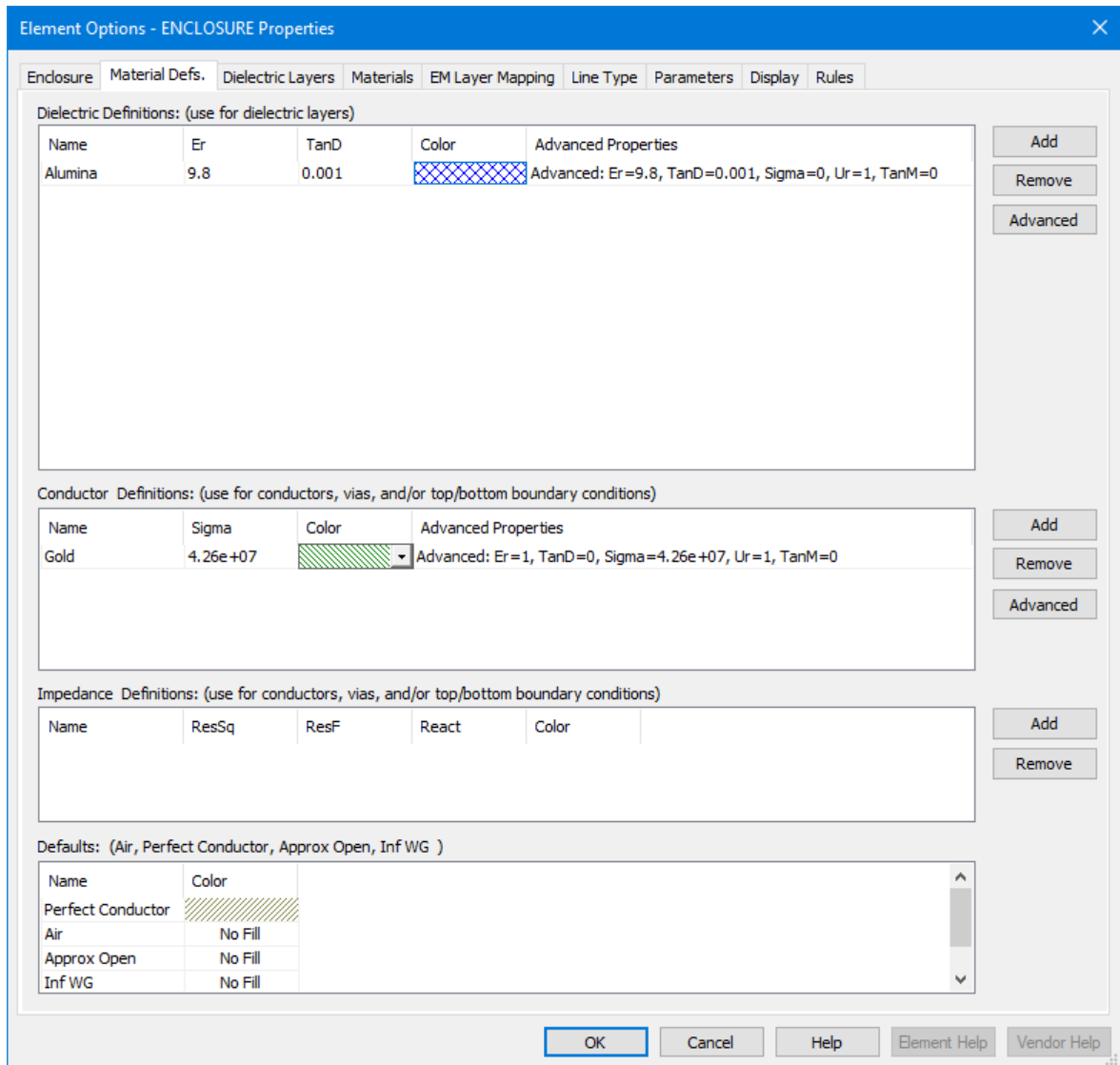


To define the materials:

1. Click the **Material Defs** tab.
2. Click the **Add** button for **Dielectric Definitions** to display the Add Dielectric dialog box. In **Preset**, choose **Alumina** from the drop-down list and set the parameters as shown in the following figure, then click **OK**.

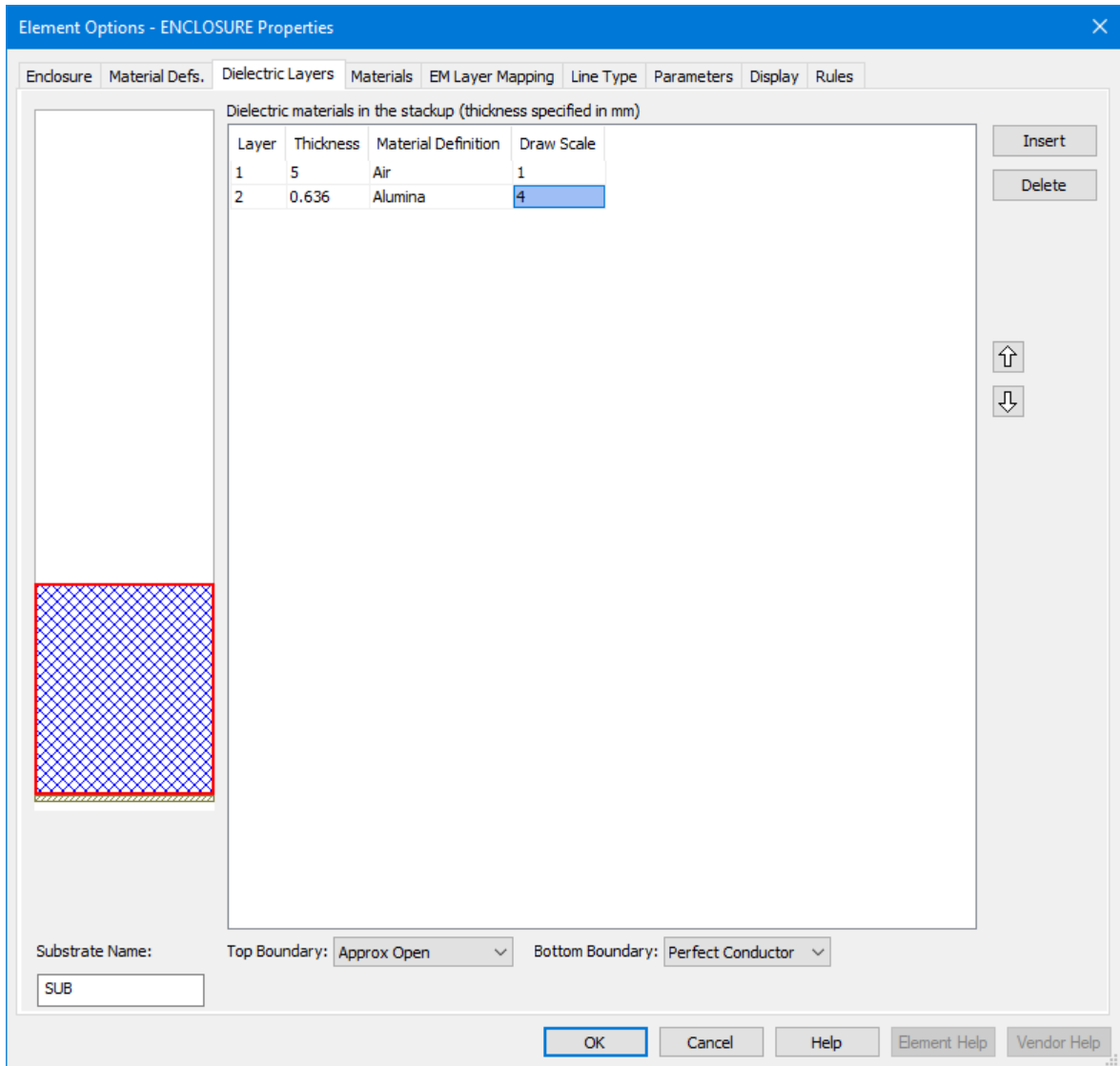


3. Click the **Add** button for **Conductor Definitions** to display the Add Conductor dialog box. In **Presets**, choose **Gold** from the drop-down list to set the parameters as shown in the following figure, then click **OK**.
4. Set the **Color** in the top two sections as shown in the following figure, or your layout colors will differ from the example colors.



To define the dielectric layers of the enclosure:

1. Click the **Dielectric Layers** tab.
2. Select **1** in the **Layer** column. Type "**5**" in the **Thickness** column, select **Air** in the **Material Definition** column, and type "**1**" in the **Draw Scale** column.
3. Select **2** in the **Layer** column. Type "**0.636**" in the **Thickness** column, select **Alumina** in the **Material Definition** column, and type "**4**" in the **Draw Scale** column.

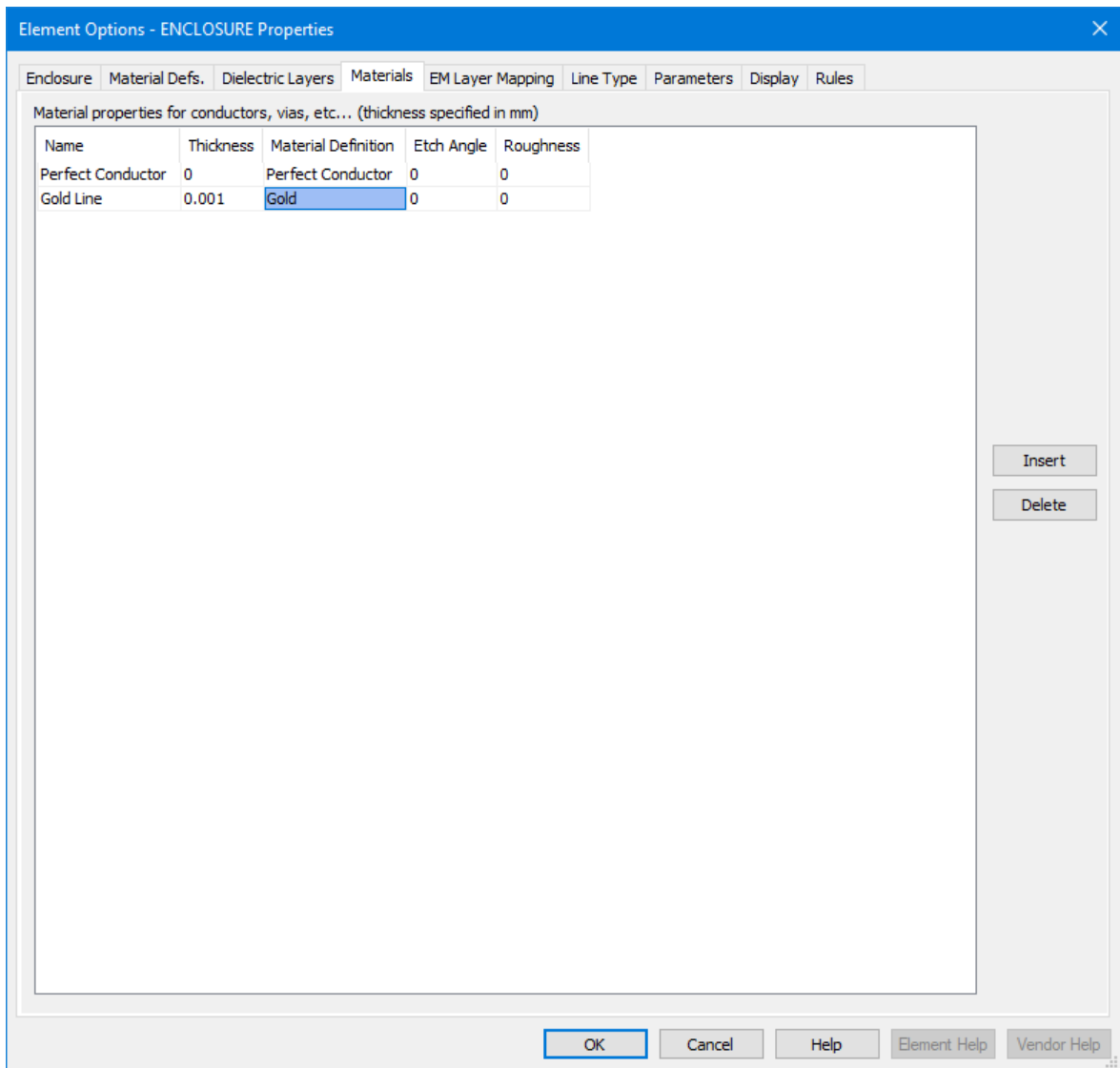


Sidewall Boundary Conditions

In AXIEM, there are no sidewall boundary conditions. Third-party simulators may make other assumptions for the sidewall boundary conditions. For details, see the vendor documentation for the solver used. The boundary conditions for the **Top Boundary** and **Bottom Boundary** of the enclosure have defaults, although you can modify these. Notice that the stackup figure in the Element Options - ENCLOSURE Properties dialog box changes when you select a different boundary condition. You do not modify the default boundary conditions in this example.

To assign material types for the conductors and vias:

1. Click the **Materials** tab in the Element Options - ENCLOSURE Properties dialog box.
2. Click the **Insert** button. In the **Name** column replace **Trace1** with "**Gold Line**", enter "**0.001**" as **Thickness**, select **Gold** in the **Material Definition** column for the 1um thick gold line, and then click **OK**.

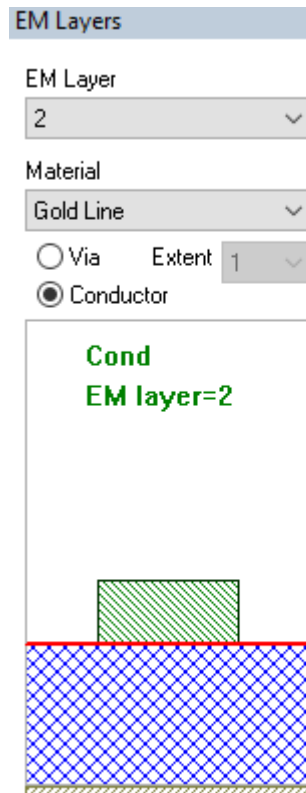


Adding Conductors to the Layout

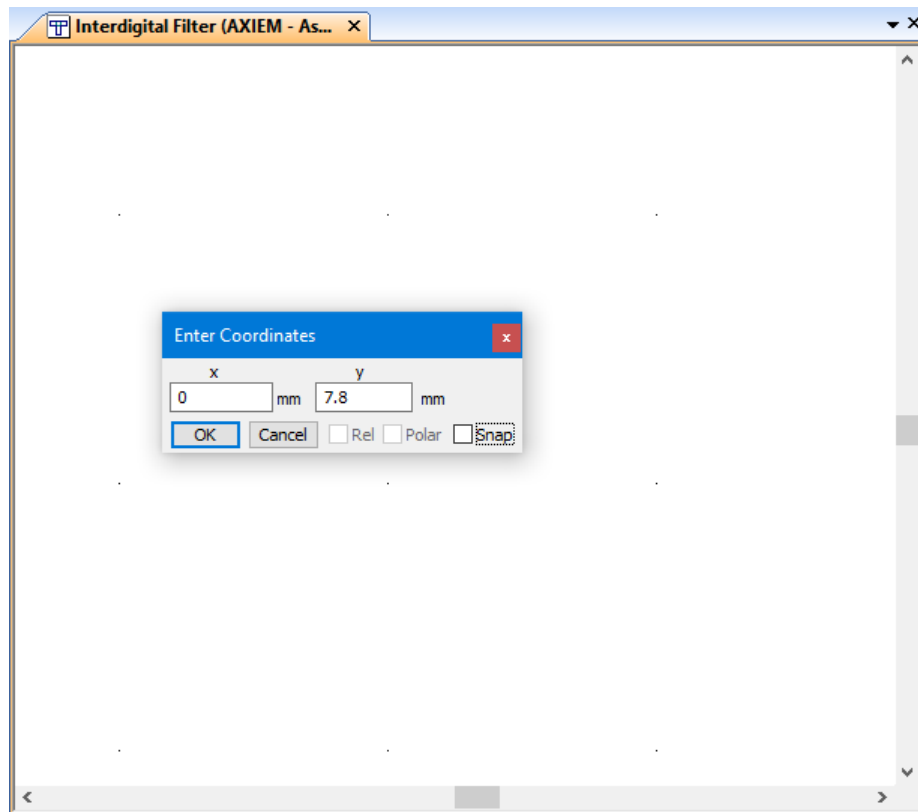
You use the Microwave Office EM Layout Editor to draw physical structures for simulation. You can also import structures directly from the NI AWR Design Environment layout tool, or import structures from AutoCAD DXF or GDSII. In this example you draw the physical layout of a microstrip interdigital filter using the EM Layout Editor.

To draw the physical layout:

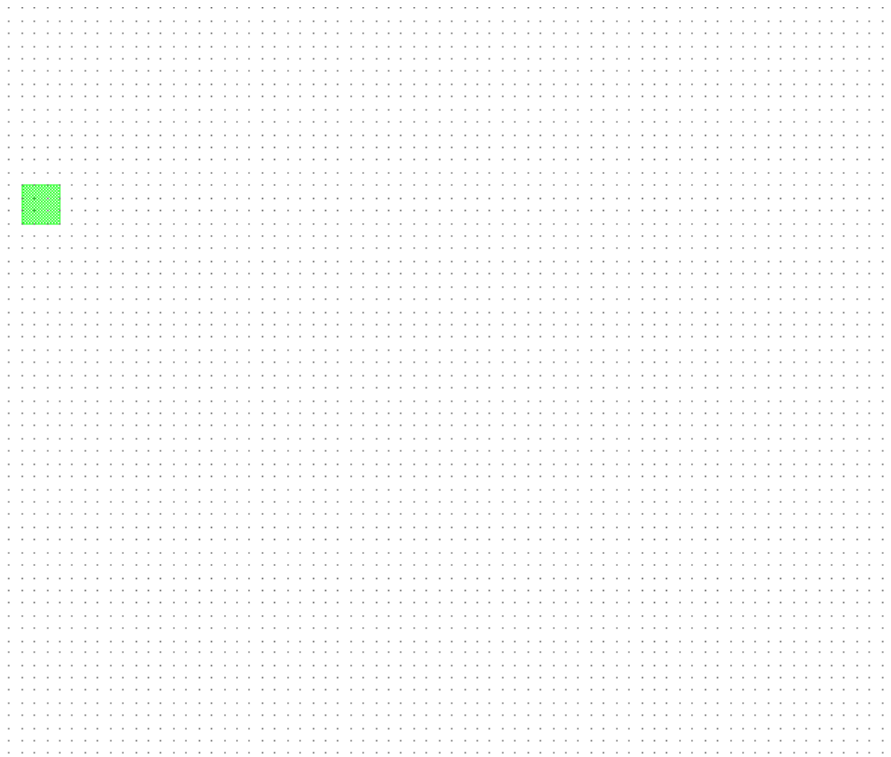
1. Click the **Layout** tab to display the Layout Manager, which is comprised of expandable sections such as EM Layers and Layout Objects. To expand or collapse a section, click the symbol on the right end of the section title bar. Expand the EM Layers section.
2. Select **2** as the **EM Layer**, **Gold Line** as the **Material**, and select **Conductor**.



3. Click the top of the Interdigital Filter window to make it active, then click the **Rectangle** button on the Draw Tools toolbar (or choose **Draw > Rectangle**) to add a rectangular conductor.
4. Move the cursor into the window and press the **Tab** key or **Space** bar. The Enter Coordinates dialog box displays for entering the coordinates at which the rectangle is placed.

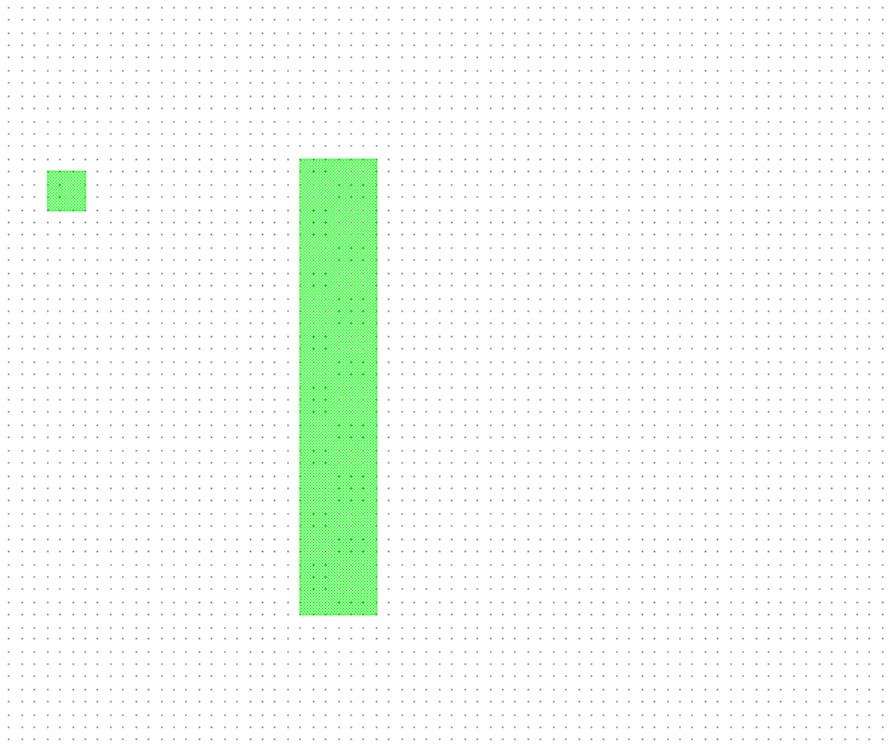


5. Type "0" as the value of **x** and "7.8" as the value of **y**, and then click **OK**.
6. Press the **Tab** key again to display the Enter Coordinates dialog box. Ensure that the **Rel** (relative) check box is selected, type "0.6" as the value of **dx**, and "0.6" as the value of **dy**, and then click **OK**. A rectangular conductor displays in the EM structure window.
7. Click the **View All** button on the toolbar (or choose **View > View All**).



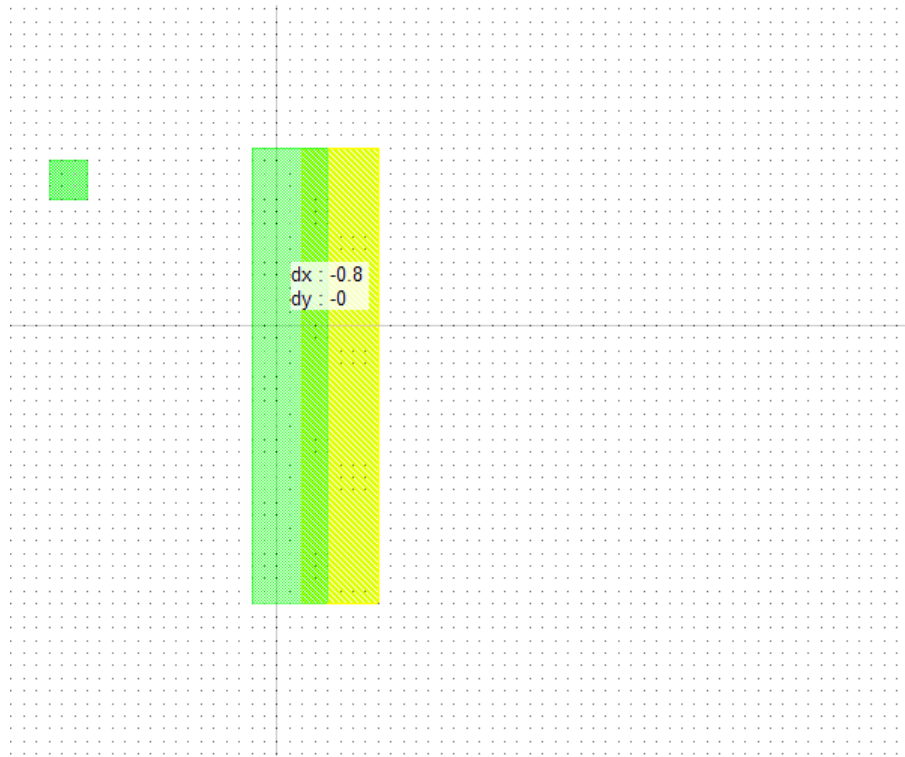
To draw a second rectangular conductor:

1. Click the **Rectangle** button on the toolbar.
2. Move the cursor into the Interdigital Filter window and press the **Tab** key. The Enter Coordinates dialog box displays. Type "4" as the value of **x** and "1.4" as the value of **y**, and then click **OK**.
3. Press the **Tab** key again to display the Enter Coordinates dialog box. Type "1.2" as the value of **dx** and "7.2" as the value of **dy**, and then click **OK**. A second rectangular conductor displays in the EM structure window.
4. Click the **View All** button on the toolbar (or choose **View > View All**).

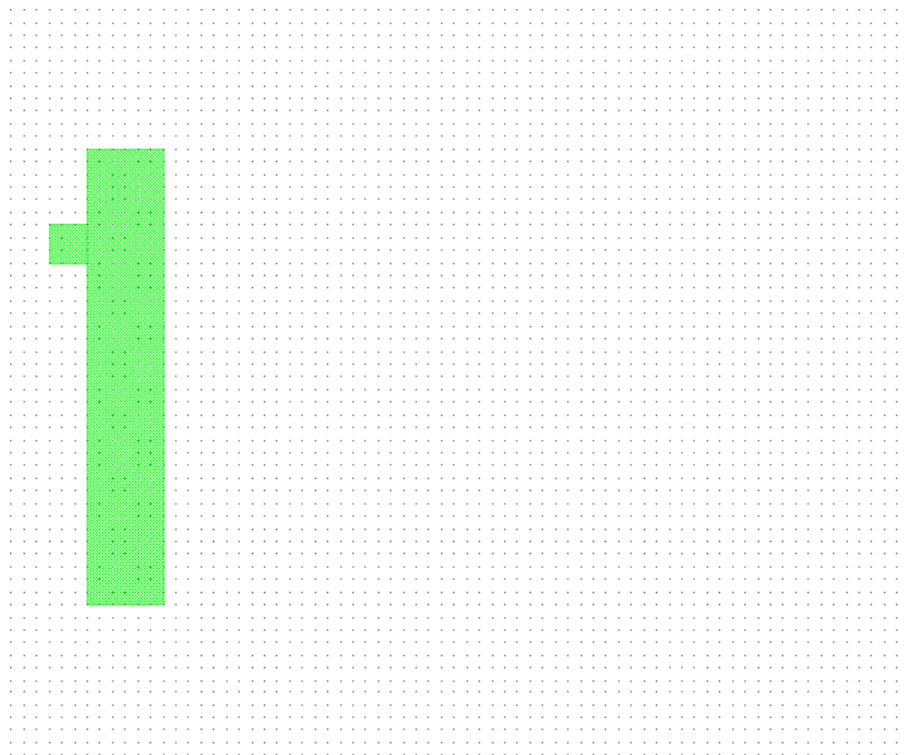


To move the second rectangular conductor next to the first conductor:

1. Click the second rectangular conductor to select it.
2. Slide the cursor over the selected conductor until the cursor displays as a cross.
3. Click and hold down the mouse button. A dx, dy readout displays in the window, as shown in the following figure.



4. Drag the cursor until the dx, dy readout displays dx: -3.4 and dy: 1, then release the button to place the rectangle.



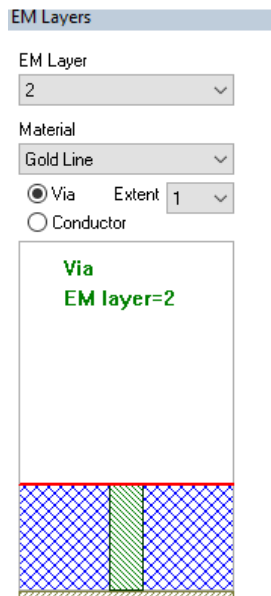
NOTE: You can click the **Measure** button on the toolbar to measure the dimension of conductors, offsets, or spaces in an EM structure layout.

Adding Vias

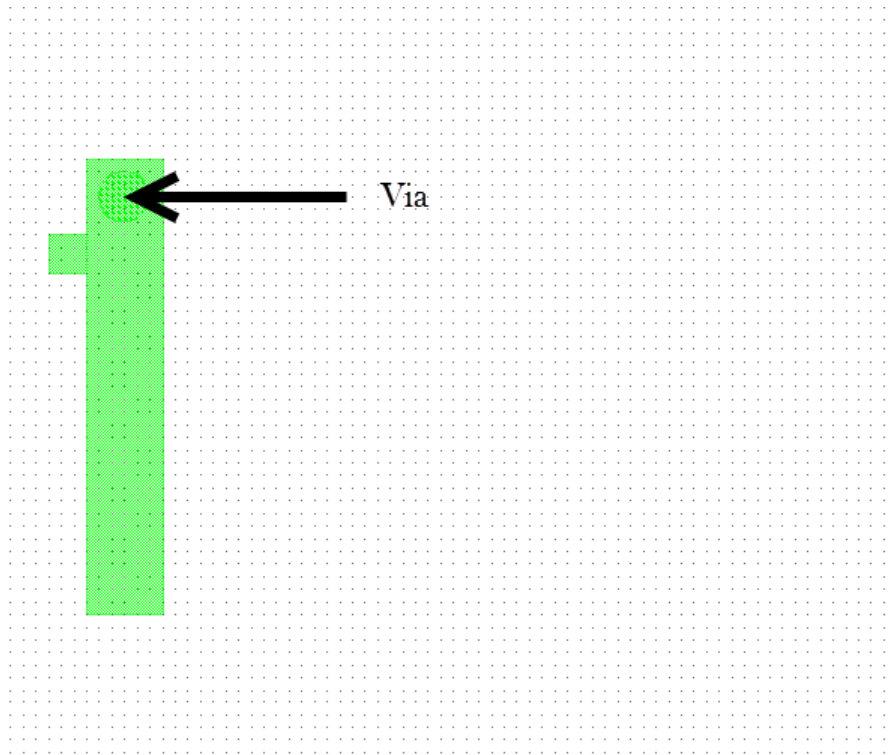
Vias are interconnects between substrate layers. You must add a via to ground from one side of the larger conductor to the bottom of the enclosure.

To add a via:

1. In the Layout Manager, select **Via** in the EM Layers section.
2. Select **Gold Line** as the **Material** and **1** as the **Extent**.



3. Choose **Draw > Ellipse**.
4. Move the cursor into the Interdigital Filter window and press the **Tab** key. The Enter Coordinates dialog box displays. Type "**0.8**" as the value of **x** and "**9.4**" as the value of **y**, and then click **OK**.
5. Press the **Tab** key again to display the Enter Coordinates dialog box. Type "**0.8**" as the value of **dx** and "**-0.8**" as the value of **dy**, and then click **OK**. A via displays in the Interdigital Filter window.



Viewing the Structure in 3D

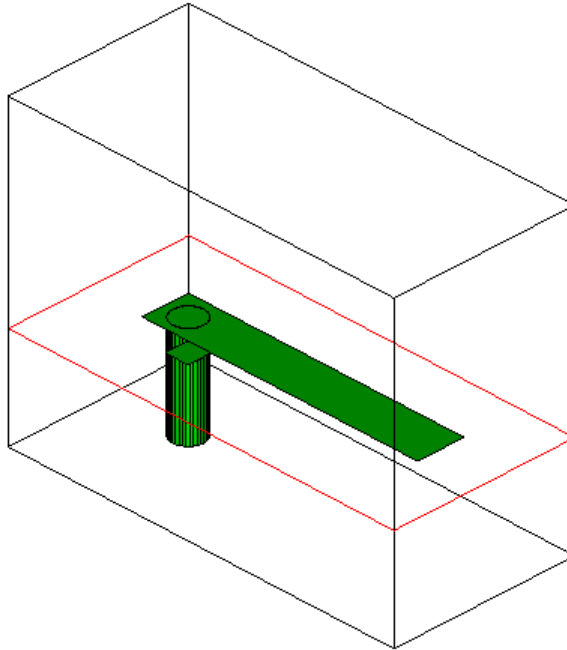
The EM simulator supports multiple 2D (structure) and 3D views.

To create a 3D view:

1. Choose **View > View 3D EM Layout**. A window containing the 3D view displays in the workspace.
2. Choose **Window > Tile Vertical**. The views display side-by-side.

NOTE: To change the view of a 3D structure, right-click in the 3D window and choose **Zoom Out**, **View Area**, or **View All**.

3. To rotate a 3D structure, click anywhere in the 3D window and hold down the mouse button while you move the mouse.



Adding Ports

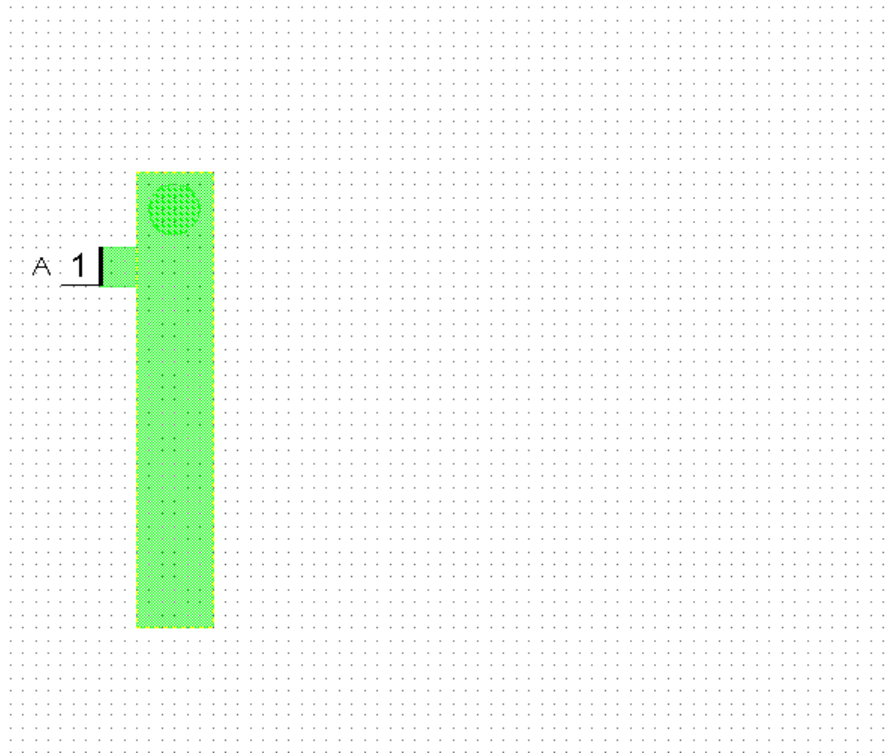
AXIEM can have electrical ports defined at the edge of drawn shapes.

To define an edge port:

1. Click the smaller conductor in the EM structure window.
2. Choose **Draw > Add Edge Port**.
3. Position the cursor to the left edge of the small conductor until the outline of a square displays, then click to place the port. A box with the number 1 (indicating port 1) displays at the left edge of the conductor.

The "A" next to the port number indicates that the port is an auto port. Auto ports automatically set the ground reference and reference plane distance for the port.

4. Click the **View All** button on the toolbar (or choose **View > View All**).



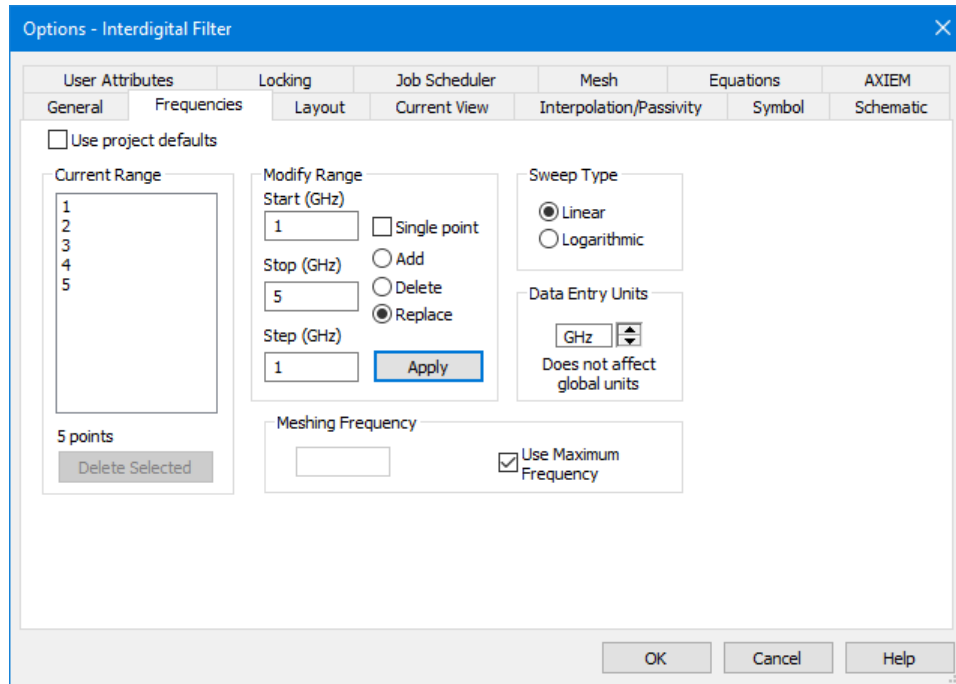
Specifying the Simulation Frequencies

To specify the simulation frequencies:

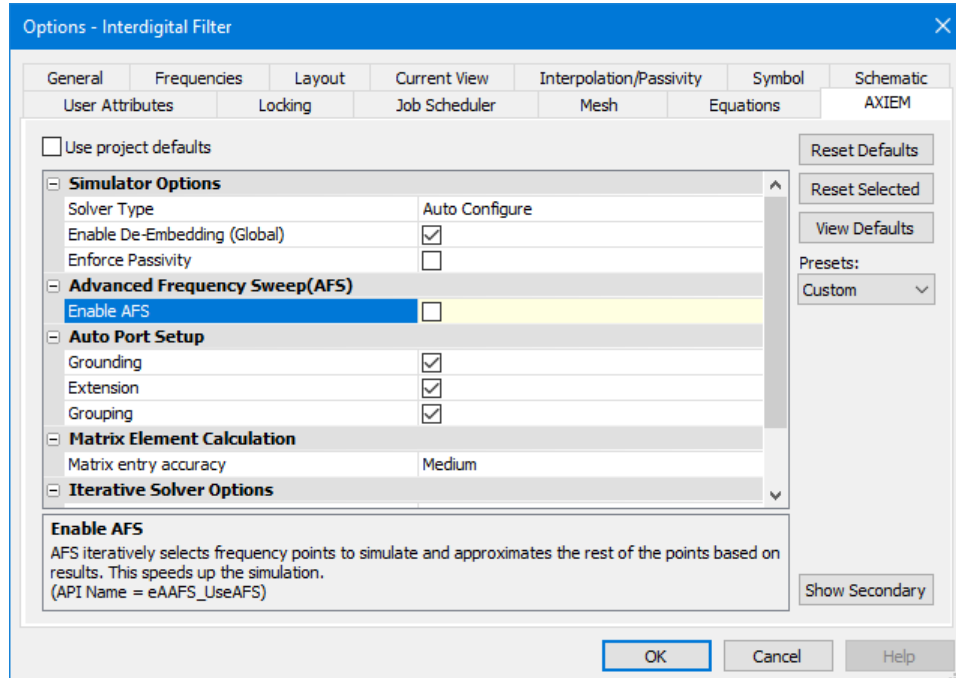
1. In the Project Browser, right-click "Interdigital Filter" under **EM Structures** and choose **Options**. The Options dialog box displays.
2. Click the **Frequencies** tab.
3. Clear the **Use project defaults** check box to give local frequency settings precedence over global project frequency settings.
4. Ensure that **GHz** displays in **Data Entry Units**.

NOTE: You can define the simulation frequency globally (by choosing **Options > Project Options** and clicking the **Frequencies** tab), or locally using these steps. It is best to use the local frequency settings for EM structures, as you typically want to sweep EM structures with fewer frequency points than with linear circuits. Data are obtained at the project frequencies using interpolation and/or extrapolation.

5. Specify the **Start**, **Stop** and **Step** values as shown in the following figure, then click **Apply** to display the values in **Current Range**.



- Click the **AXIEM** tab and clear the **Enable AFS** check box under **Advanced Frequency Sweep (AFS)** as shown in the following figure, and then click **OK**.

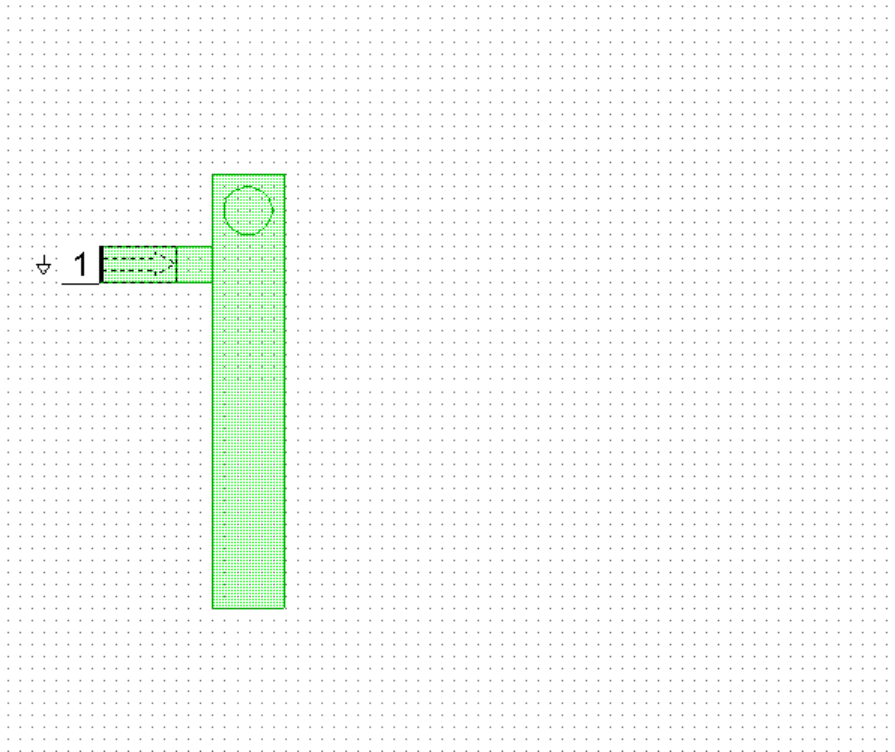


Previewing the Geometry

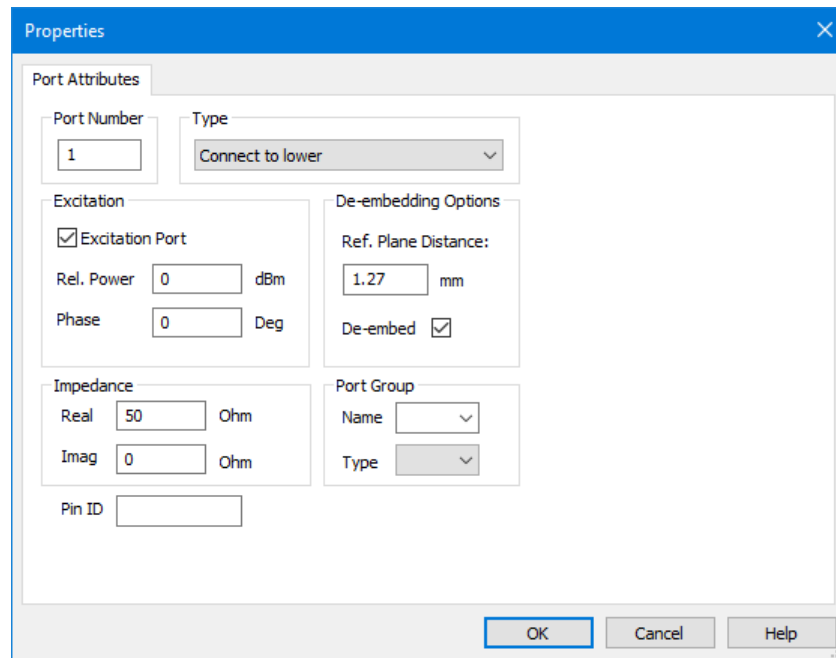
A preview of the geometry of an EM structure shows the structure after the geometry is simplified by any geometry simplification rules that it has, and also shows the grounding reference(s) and reference plane distance the auto port is using.

To preview the geometry:

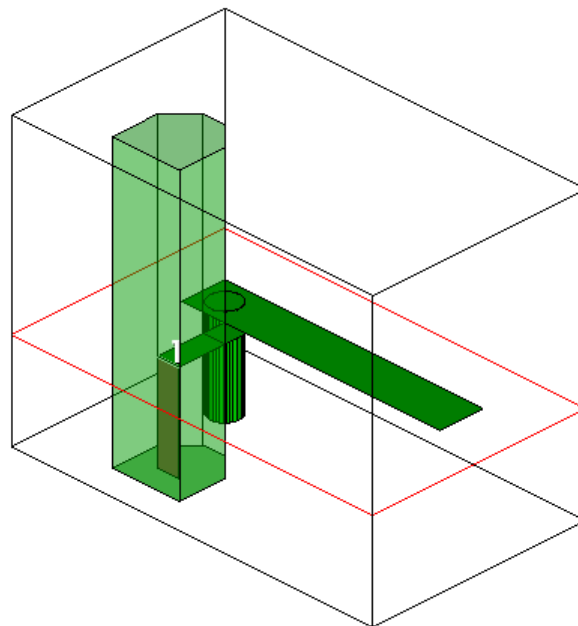
1. In the Project Browser, under the **EM Structures** node, right-click "Interdigital Filter" and choose **Preview Geometry**. The 2D layout view of the structure preview displays.



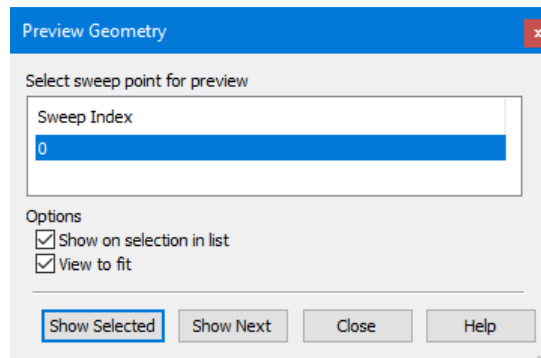
2. Double-click the port to display the Properties dialog box. The **Type** is set to **Connect to lower**, and the **Ref. Plane Distance** is set to "1.27" mm.



3. Choose **View > View 3D EM Layout**. A 3D view window displays in the workplace.
4. The reference plane extension and explicit grounding strap are visible.



5. Close the geometry preview by clicking the **Close** button on the Preview Geometry dialog box.

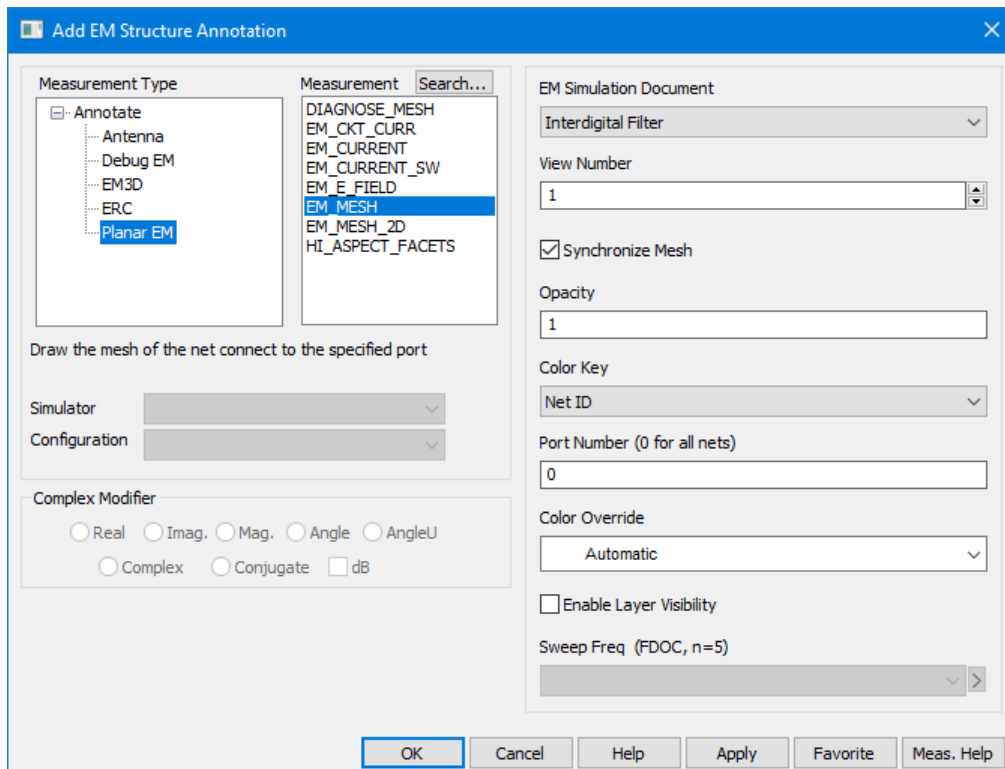


Viewing Structure Mesh

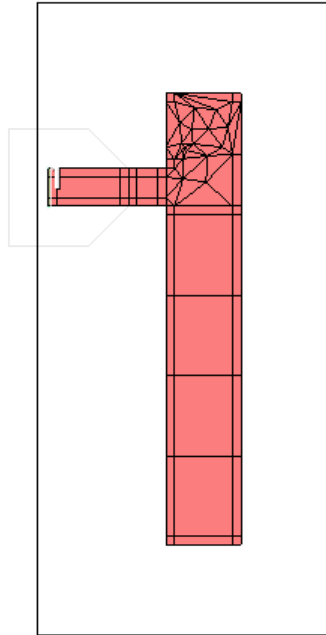
Meshing of a structure is very important before simulation. The style of mesh largely affects the simulation time. Loosely meshed structures take much less time to simulate than tightly meshed structures. Because the results may not be very different, you should start with the coarsest possible grid and gradually make it finer until the result does not change significantly. You can view mesh by adding an annotation to the structure; it displays in the 3D View.

To view mesh:

1. In the Project Browser, right-click "Interdigital Filter" under **EM Structures** and choose **Add Annotation**. The Add Annotation dialog box displays.
2. Select **EM_MESH** from the **Measurement** list and set the parameters as shown in the following figure, then click **OK**.



3. Right-click "Interdigital Filter" in the Project Browser and choose **Mesh**.
4. Make the 3D view window active. Choose **View > View From > Top**, or click the **Top** button on the toolbar for the top view, and the mesh displays as follows.



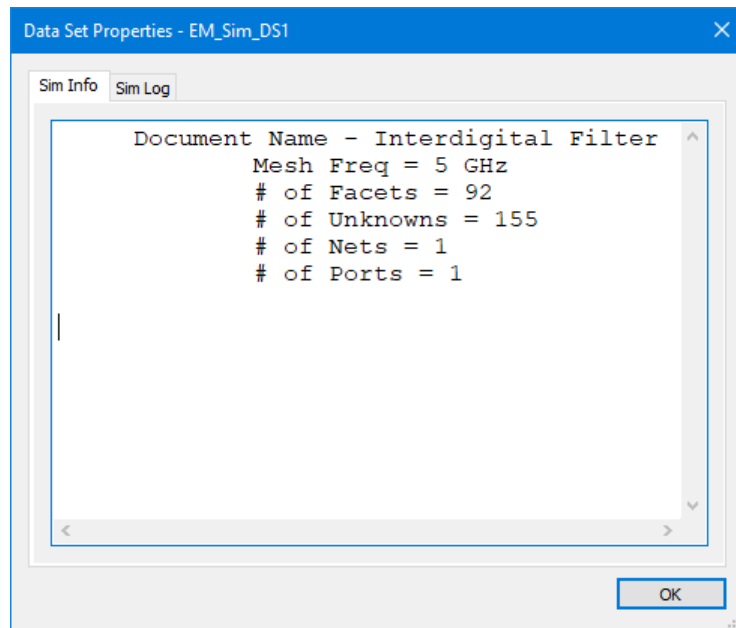
5. You can also see the reference plane extension and grounding strap in the meshed structure.

Running the EM Simulator

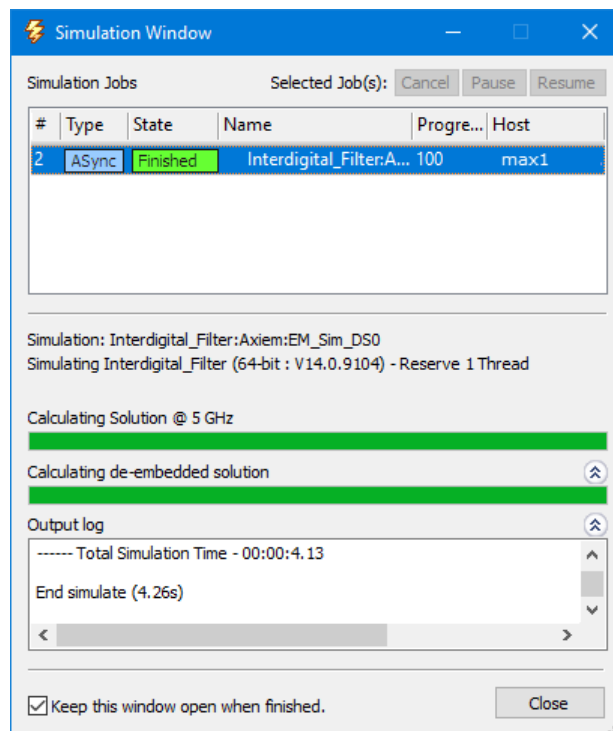
The EM simulator is very fast for electrically small structures. To find the resonant frequency of the first resonator of the filter, you can run an EM simulation on the initial layout of the Interdigital Filter EM structure.

To simulate the structure:

1. In the Project Browser, under the **EM Structures** node, under "Interdigital Filter", double-click **Information**. A Data Set Properties dialog box displays similar to the following. The maximum number of unknowns for the EM structure and other information displays, depending on the selected solver. For more information about the data displayed for third-party simulators, see the associated vendor documentation.



2. Click **OK** to close the dialog box.
3. Choose **Simulate > Analyze**. A Simulation dialog box displays to indicate the simulation progress. You can monitor progress in the **Output log** section of the dialog box. An "Estimated Time to Completion" message provides a good estimate of time remaining before completion when AFS (Advanced Frequency Sweep) is not enabled. This information is not available when using AFS, as the number of additional frequencies required for AFS convergence is unknown.

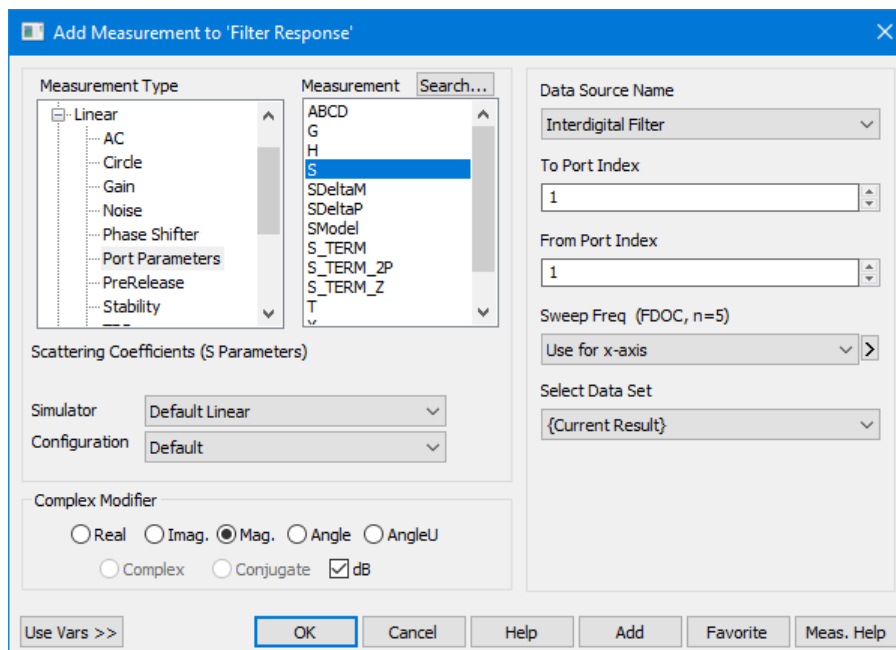


Displaying Results on a Graph

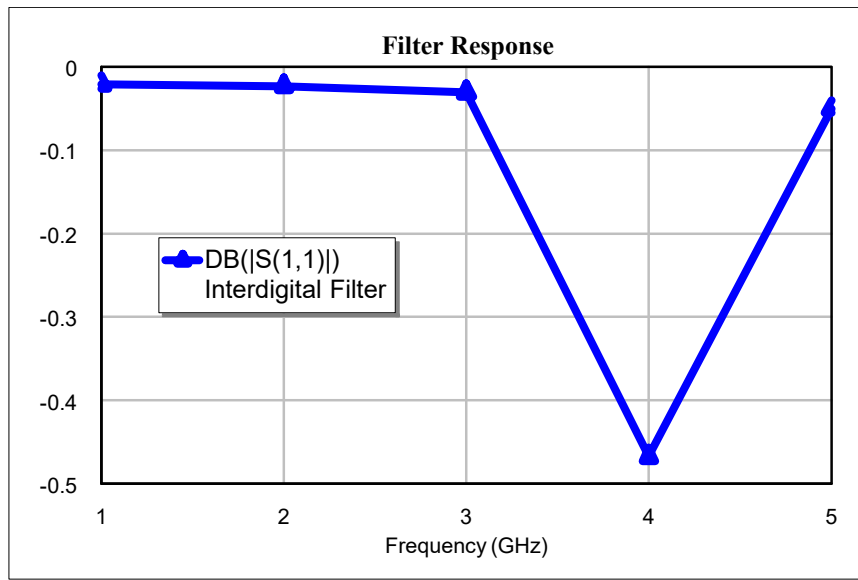
To determine the resonant frequency, you must plot the return loss of the EM structure.

To measure the resonant frequency on a graph:

1. In the Project Browser, right-click **Graphs** and choose **New Graph**. The New Graph dialog box displays.
2. Type "**Filter Response**" as the graph name and select **Rectangular** as the graph type, then click **Create**. The graph displays in the workspace.
3. Right-click the "Filter Response" graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays.
4. Create a measurement using the settings in the following figure, then click **OK**.



5. Choose **Simulate > Analyze**. The simulation response in the following graph displays. The measurement indicates that the resonant frequency is at 4 GHz.

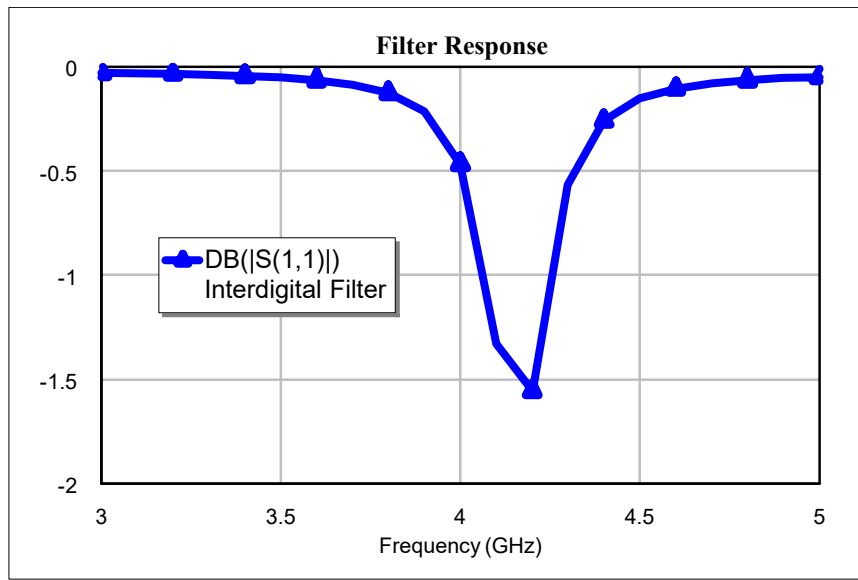


To determine a more precise measurement of the resonant frequency, you must change the frequency range and step size of the simulation.

Changing Frequency Range and Step Size

To change the simulation frequency range and step size:

1. In the Project Browser, right-click "Interdigital Filter" under **EM Structures** and choose **Options**. The Options dialog box displays.
2. Click the **Frequencies** tab.
3. Type "3" in **Start**, type "5" in **Stop**, and type "0.1" in **Step**. Select **Replace** and click the **Apply** button, then click **OK**.
4. Choose **Simulate > Analyze** to re-analyze the circuit. The simulation response in the following graph displays.



Animating Currents

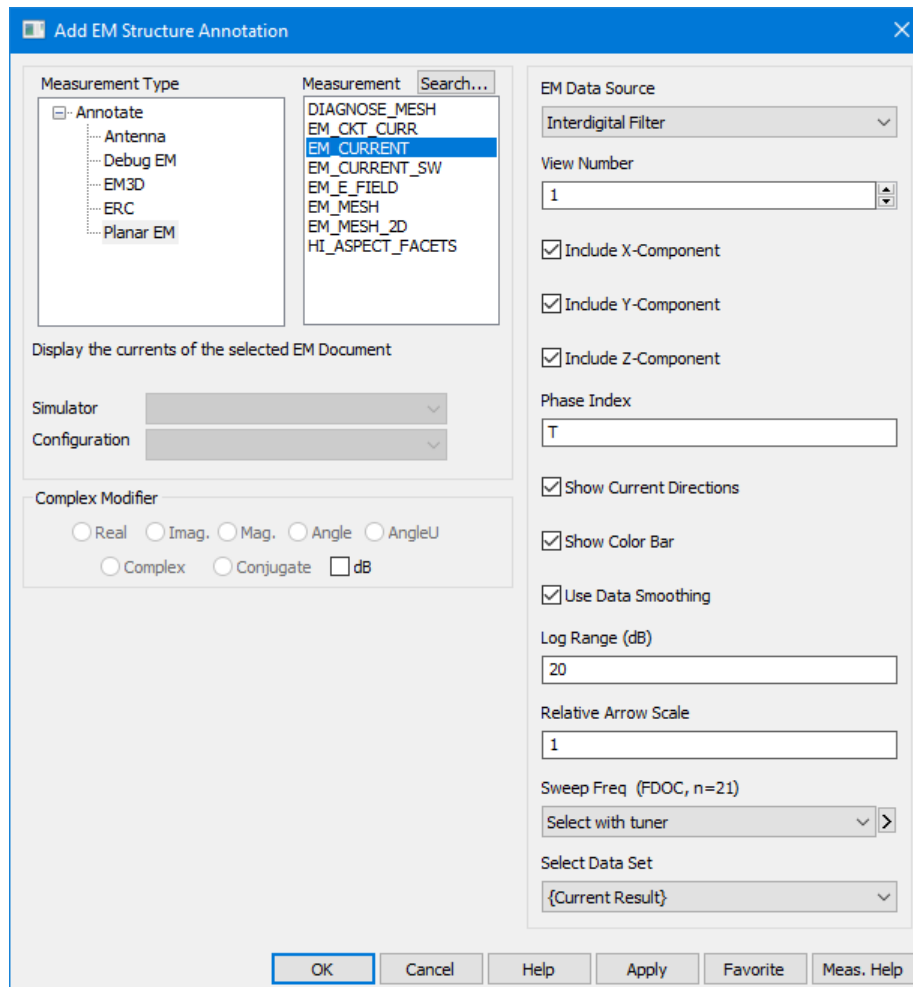
Viewing the currents of an EM structure can be useful when studying its physical characteristics. Currents are added as annotations to the EM structure and are displayed in a 3D view. For details on imaging when using an alternate simulator, see the associated vendor documentation.

To enable current display, right-click "Interdigital Filter" and choose **Options** to display the Options dialog box. Click the **General** tab and select the **Currents** check box under **Save Results in Document**, then click **OK**.

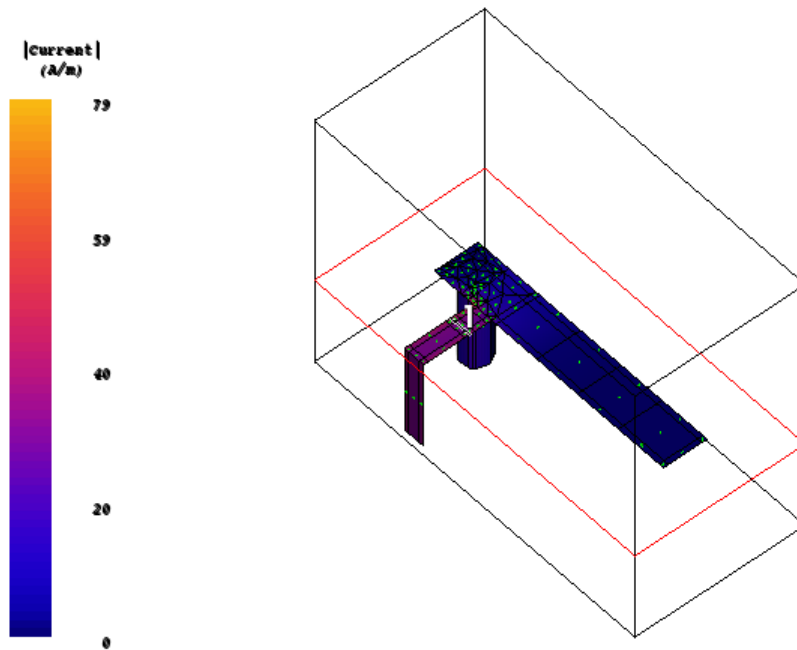
Because you did not request currents in the previous simulation, you cannot plot EM currents from the previously simulated data set. A new simulation is required to calculate the currents. To resimulate, right-click "Interdigital Filter" and choose **Force Re-simulation**, then choose **Simulate > Analyze** to resimulate.

To animate the currents on the conductors:

1. In the Project Browser, right-click "Interdigital Filter" under **EM Structures** and choose **Add Annotation**. The Add EM Structure Annotation dialog box displays.
2. Select **EM_CURRENT** from the **Measurement** list and set the parameters as shown in the following figure, then click **OK**.

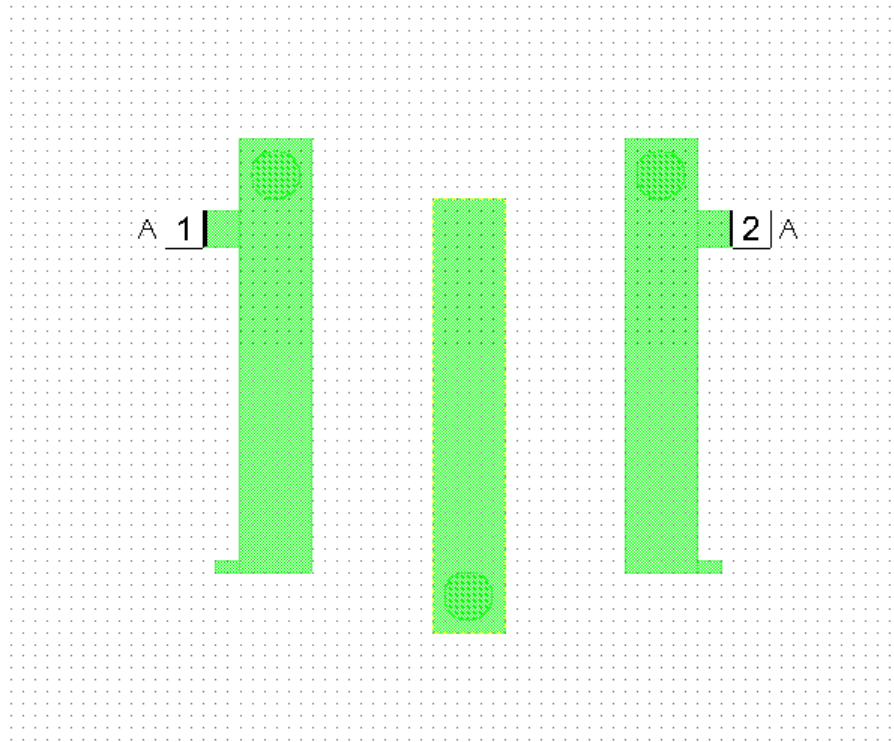


3. Click the 3D window of the Interdigital Filter EM structure to make it active.
4. Click the **Animate Play** button on the EM 3D Layout toolbar to animate the current in the 3D view as shown in the following figure.
5. Click the **Animate Stop** button on the toolbar to stop the animation. Click on the structure and move the cursor to rotate the view.



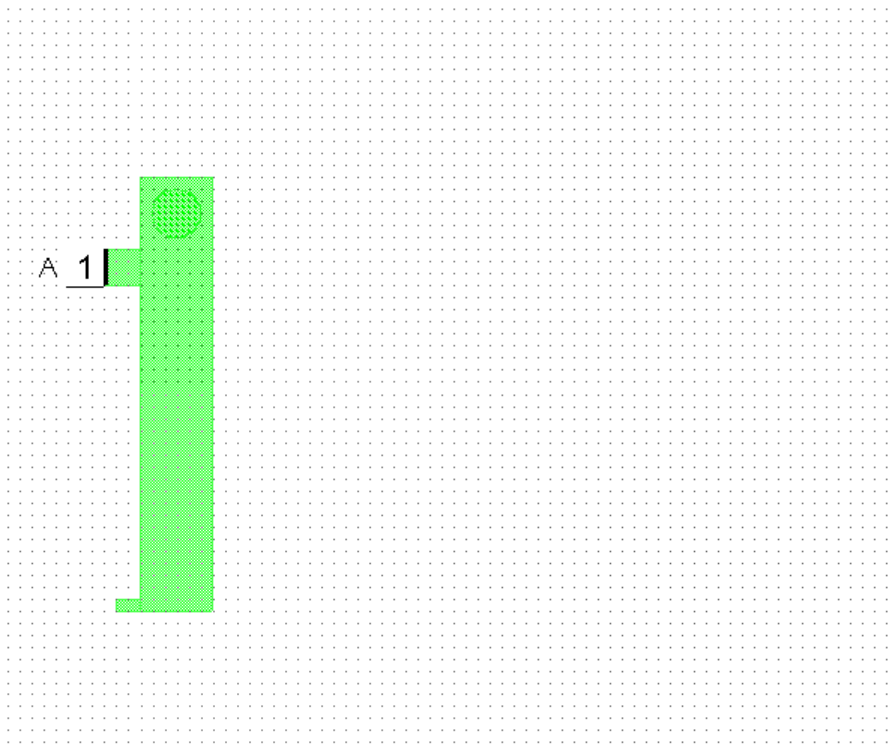
Completing the Filter Layout

To complete the following filter you use some advanced editing features in the EM structure window.



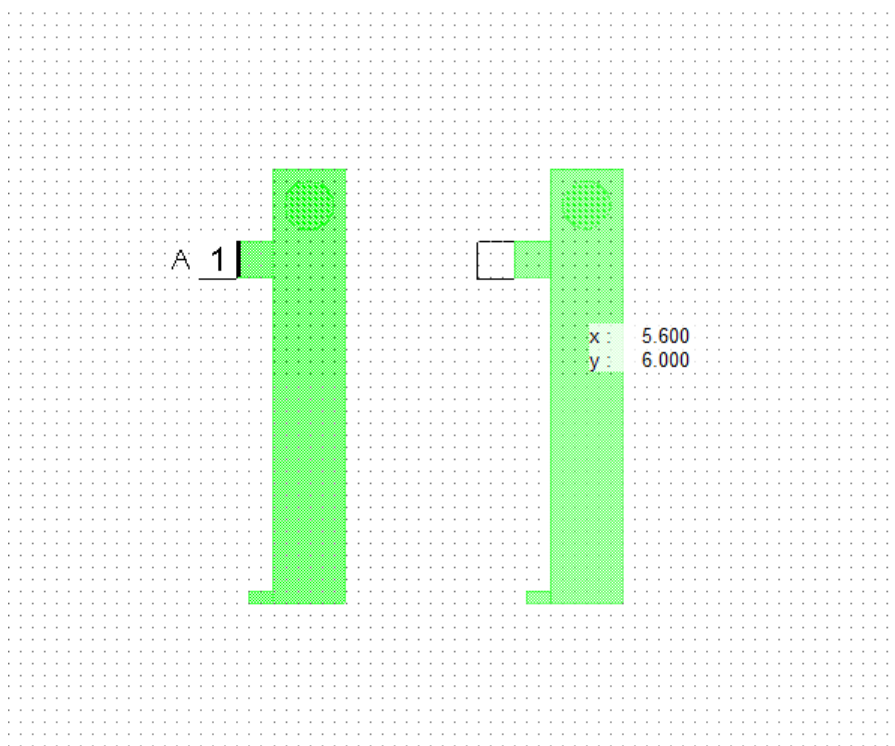
To add a small conductor at the end of the input resonator:

1. Click the **Layout** tab and select **Conductor**.
2. Select **2** as the **EM Layer**, and **Gold Line** as the **Material**.
3. Choose **Draw > Rectangle**, and press the **Tab** key to display the Enter Coordinates dialog box.
4. Type "**0.6**" as the **x** value and "**2.4**" as the **y** value, then click **OK**.
5. Press the **Tab** key again to display the Enter Coordinates dialog box. Select **Rel**. Type "**-0.4**" as the **dx** value and "**0.2**" as the **dy** value, then click **OK**. A rectangular conductor displays in the EM structure window.

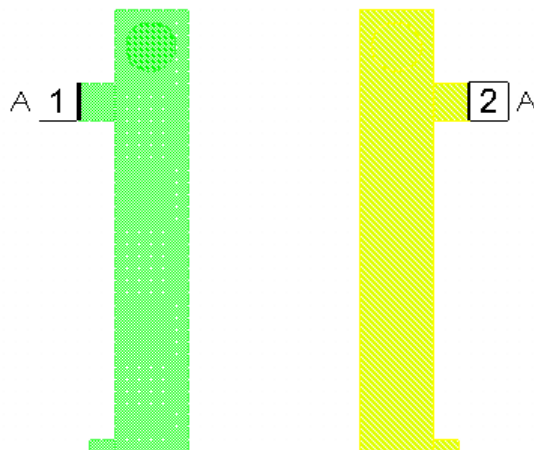


To draw the output resonator:

1. Choose **Edit > Select All**.
2. Choose **Edit > Copy**, and then choose **Edit > Paste**. An outline of the input resonator displays.



3. Move the cursor to the right. **Ctrl-right-click** once to flip the selected instance.



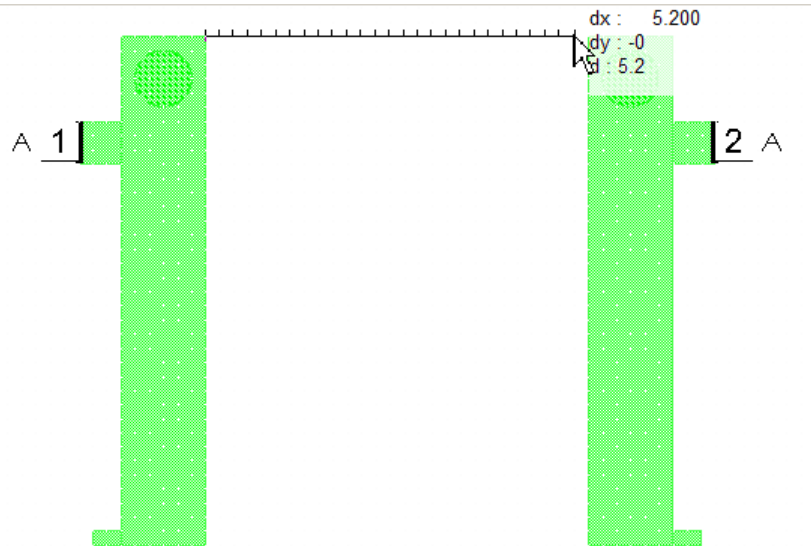
4. Click the **View All** button on the toolbar (or choose **View > View All**).

To move the flipped instance:

1. While still selected, drag the instance until the distance between the two instances equals 5.2 mm.

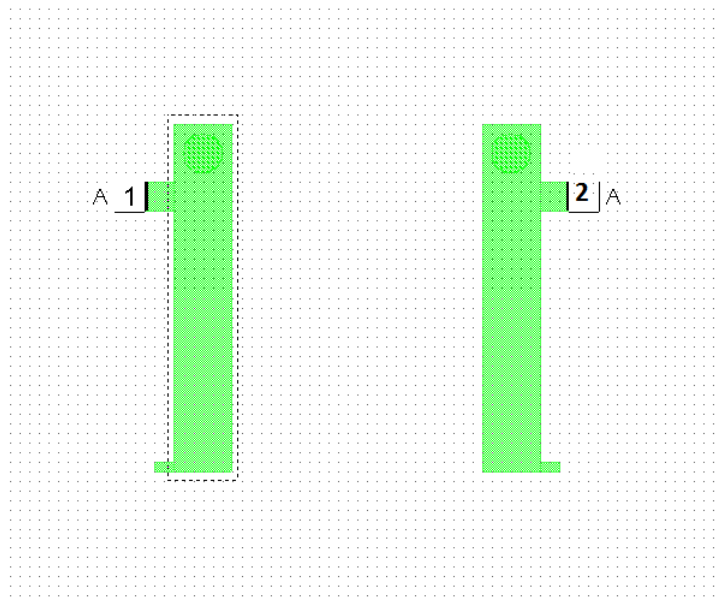
NOTE: To change the view, right-click and choose **Zoom Out**, **View Area**, or **View All**.

2. To measure the distance between two points, choose **Draw > Measure** or click the **Meas Tool** button on the toolbar, then click on the first node and slide the cursor to the second point.



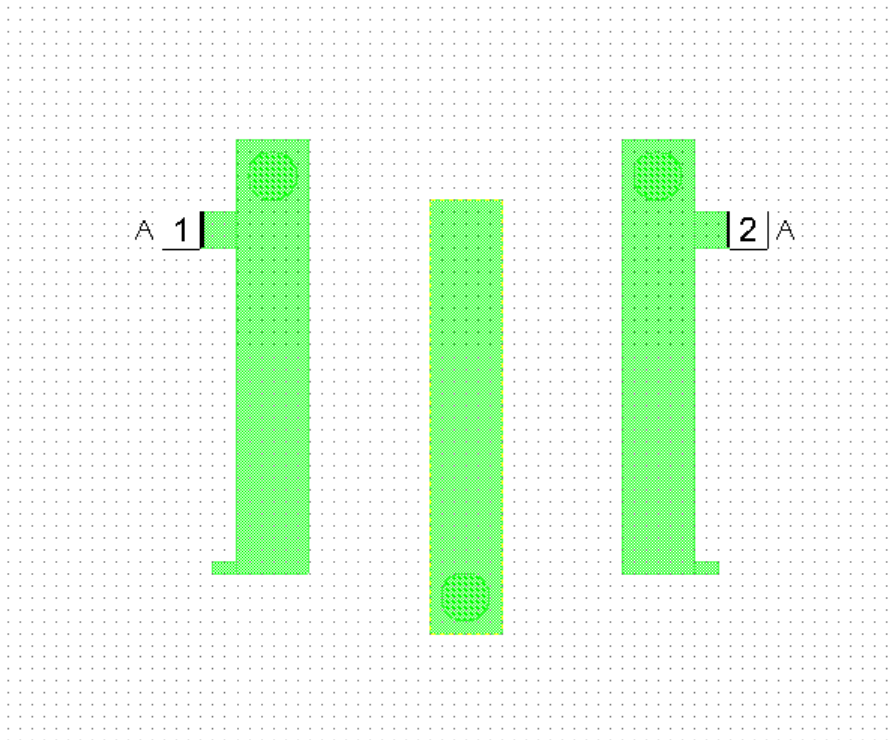
To create the middle resonator:

1. Click near the top left corner of the left-most resonator, hold down the mouse button, and drag the cursor down and to the right so the dashed box encompasses the resonator, then release the mouse button. The large conductor and the via are selected.



2. Choose **Edit > Copy** and then **Edit > Paste**. An outline of the copied instance displays.

3. Move the cursor to the middle of the EM structure window to move the copied instance, then right-click twice to rotate the instance 180-degrees.
4. With the copied instance still highlighted, move the cursor to place it directly on top of the original input resonator, then press the **Tab** key.
5. Clear **Rel** in the Enter Coordinates dialog box and type "4.4" as the **x** value and "5" as the **y** value, then click **OK**. The EM Structure displays as follows.

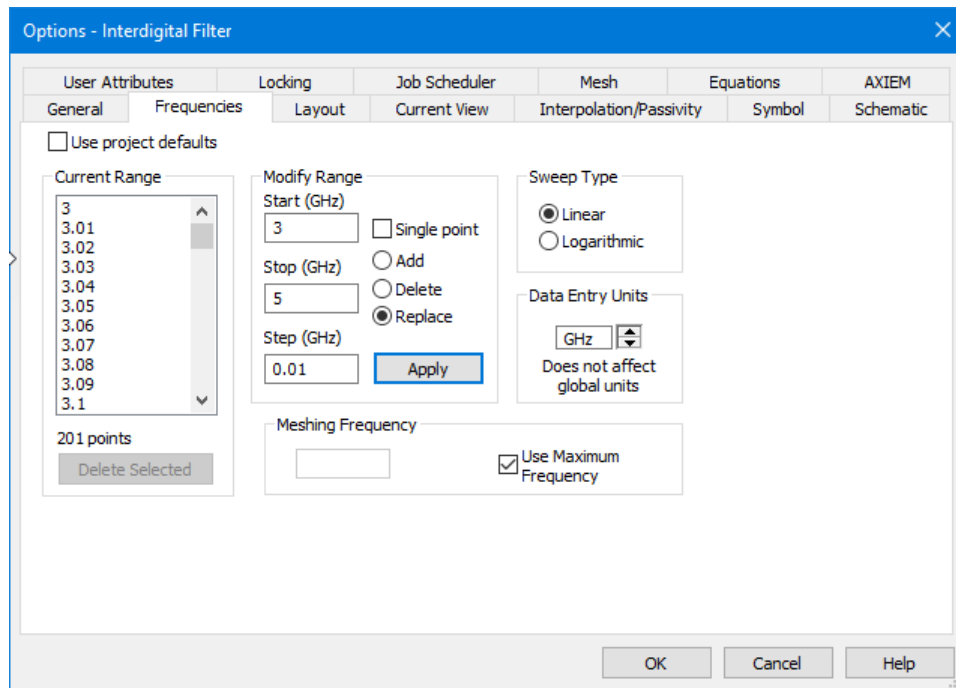


Advanced Frequency Sweep

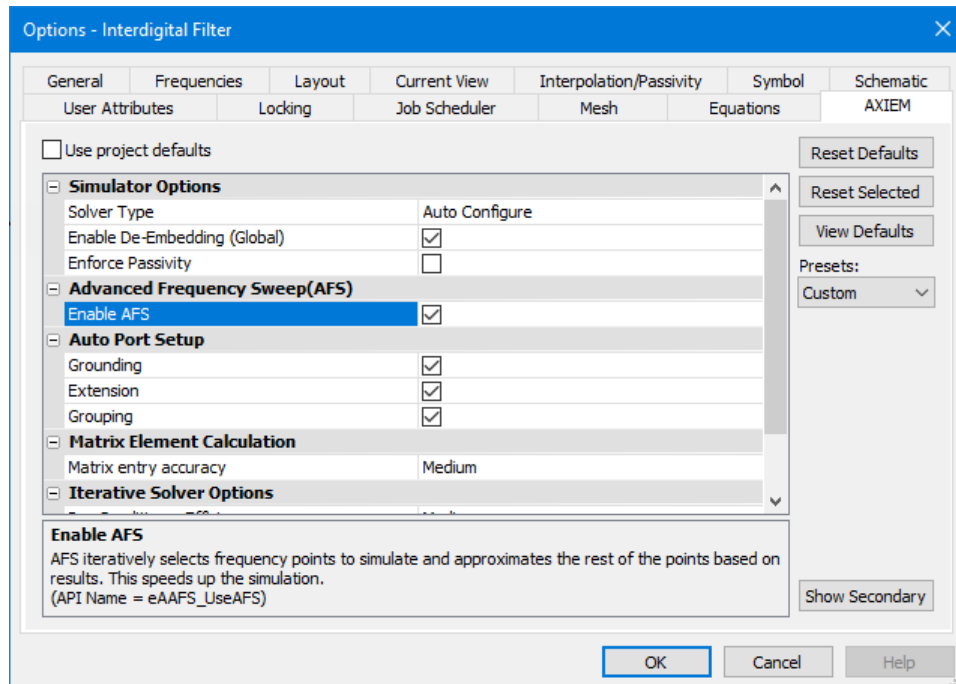
AXIEM can perform Advanced Frequency Sweep (AFS), which speeds a simulation by allowing the simulator to determine the frequencies needed to obtain an accurate response. Note that the currents are not calculated while performing AFS.

To perform an advanced frequency sweep:

1. In the Project Browser, right-click "Interdigital Filter" under **EM Structures** and choose **Options**.
2. Click the **Frequencies** tab and specify the **Start**, **Stop**, and **Step** values as "3", "5", and "0.01" respectively. Select **Replace** and click the **Apply** button to display the values in **Current Range**.

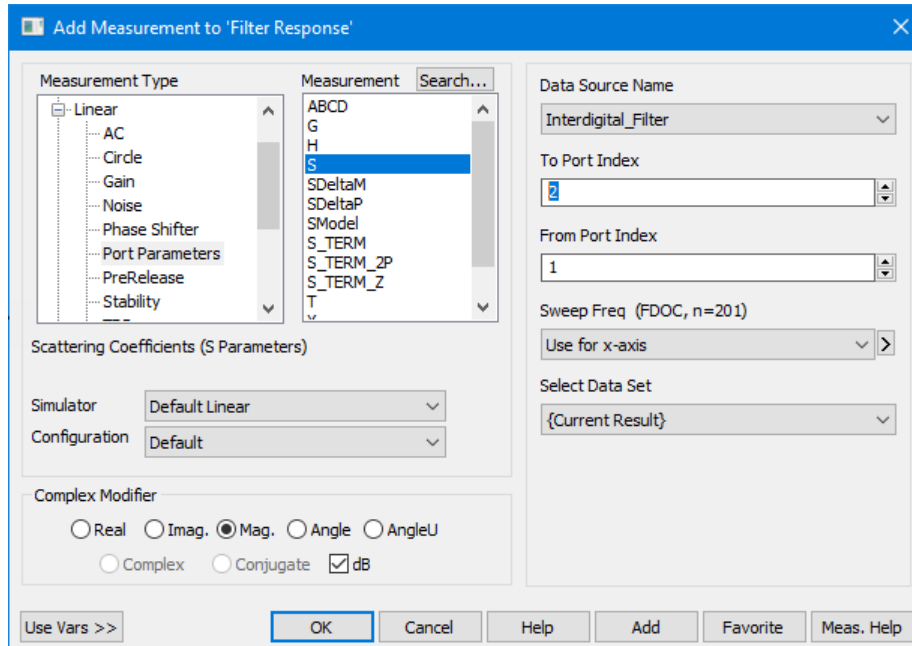


3. Click the **AXIEM** tab and select the **Enable AFS** check box under **Advanced Frequency Sweep (AFS)** as shown in the following figure, then click **OK**.

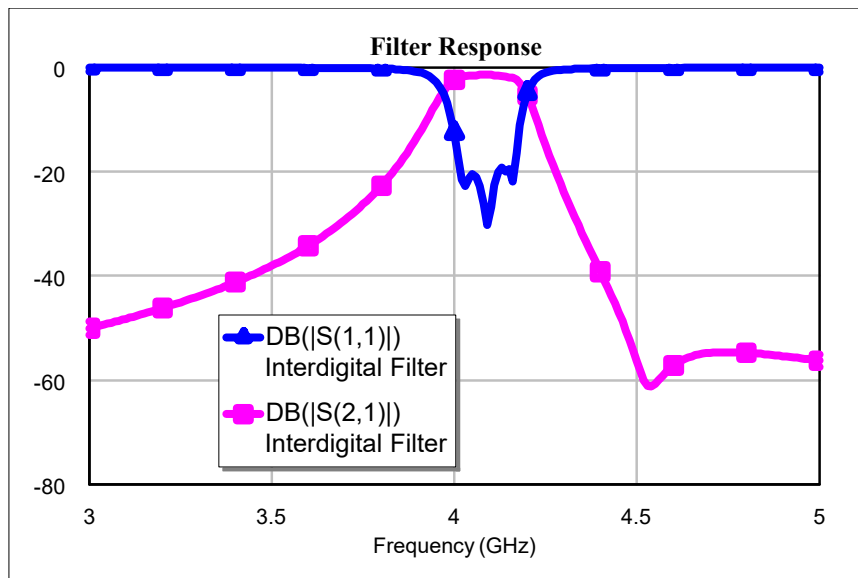


4. Right-click the "Filter Response" graph in the Project Browser and choose **Add Measurement**. The Add Measurement dialog box displays.

5. Create a measurement using the settings in the following figure, then click **OK**.



6. Choose **Simulate > Analyze**. The final simulation response on the following graph displays.

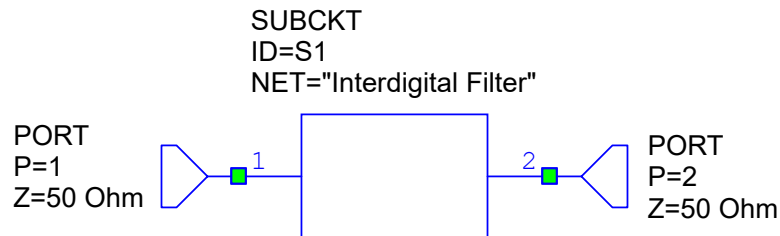


If the EM structure 3D window is open, an error displays relating to using AFS and the current annotation. In the Status Window, click the **Warnings** button. A warning displays with a link you can click for help.

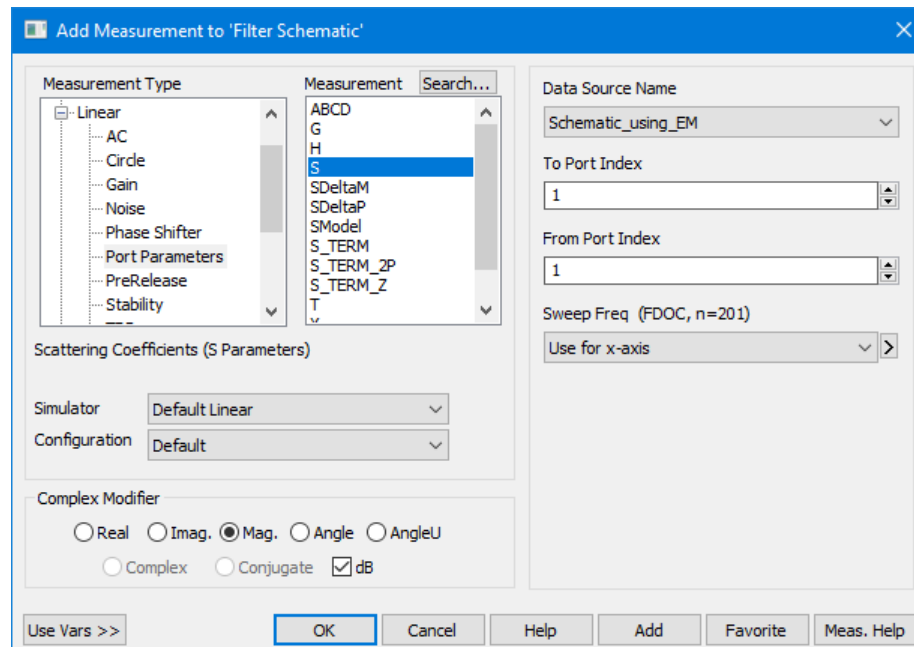
Adding an EM Structure to a Schematic as a Subcircuit

You can add an EM structure to a schematic in the same way you add a schematic subcircuit. The EM structure added as a subcircuit has the same number of ports as the structure.

1. Create a new schematic named "Schematic using EM" (choose **Project > Add Schematic > New Schematic** or click the **Add New Schematic** button on the toolbar).
2. Click the **SUB** button on the toolbar or choose **Draw > Add Subcircuit** to add a subcircuit. A window displays with the EM structure name highlighted.
3. Click **OK**. Drag the subcircuit onto the schematic and click to place it.
4. Click the **Port** button on the toolbar to add two ports to the subcircuit as shown in the following figure.



5. Right-click "Schematic using EM" under **Circuit Schematics** in the Project Browser and choose **Options** to display the Options dialog box.
6. Click the **Frequencies** tab and clear the **Use project defaults** check box.
7. Type "3" in **Start**, type "5" in **Stop**, and type "0.01" in **Step**. Click the **Apply** button, and then click **OK**.
8. Add a rectangular graph named "**Filter Schematic**".
9. With the "Filter Schematic" graph window active, click the **Add New Measurement** button on the toolbar and create a measurement using the settings in the following figure, then click the **Apply** button.



10. In the Add Measurement dialog box, change **To Port Index** to **2** and click **OK**.
11. Click the **Analyze** button on the toolbar to simulate the schematic and compare the two graphs. The results should match.
12. Save and close the project.

Index

Symbols

3D structures, 7–16

A

Adding

- data files to schematics, 3–4
- measurements, 2–18
- ports, 2–11
- schematic back annotation, 6–8
- subcircuits to diagrams, 2–11
- subcircuits to schematics, 2–11

Analyzing

- a circuit, 4–9

Anchoring layout cell, 5–12

Annotation

- back, 6–8

Artwork cells

- adding ports, 5–16
- assigning to schematic element, 5–9
- creating, 5–13

B

Back annotation, 6–8

Basic operations, 2–4

Bias circuit, 6–6

C

Cell

- libraries, 2–16

Circuit

- analyzing, 4–9
- DC bias, 6–6
- optimizing, 4–15
- power amplifier, 6–1
- tuning, 4–11

Command

- shortcuts, 2–19

Component Browser, 6–21

Conductor

- adding to layout, 7–9

Connecting nodes, 2–12

Connectivity Checker, 5–10

Conventions; typographical, 1–2

Copying

- schematics, 6–23

Creating

- layout, 2–14

Current

- animation, 7–27

Curve meter, 6–4

D

Data files

- adding to schematic, 3–4
- importing, 3–1, 5–4
- placing in schematic, 3–4
- Touchstone, 3–1

Database units

- default grid size, 5–3

Default

- project units, 4–1

Distributed interdigital filter, 7–2

Documentation; NI AWR, 1–3

Dynamic load line measurement, 6–21

E

Electromagnetic (EM) simulator, 7–1

Element Browser, 2–4, 2–9

Element symbol; changing, 5–5

Elements

- adding to schematics, 2–9, 4–2

EM

- simulation, 7–1

EM structures

- adding to schematic, 7–37
- creating, 2–12, 7–3
- drawings, 2–13
- enclosure, 7–4

Enclosure

- properties, 7–4

Examples

- opening, 2–5

F

Filters

- distributed interdigital, 7–2
- layout, 7–29

Frequency

- setting, 4–6
- simulation, 3–6

G

GDSII

- cell library, 5–4

Geometry

- preview, 7–20

Graph

- adding measurements, 2–17
- creating, 2–17
- displaying results on, 7–25
- types, 2–17

Ground node; adding, 4–5, 5–4

H

Harmonic balance, 6–1

- port, 6–10, 6–11
- single-tone analysis, 6–1

Help

- online, 1–5, 2–20

Hotkeys, 2–19

I

Importing

- data files, 3–1, 5–4
- GDSII cell library, 5–4
- layer process file (LPF), 5–2
- S-parameter files, 3–1

K

Keyboard shortcuts, 2–19

Knowledge Base; NI AWR, 1–3

L

Large signal reflection coefficient, 6–13

Layer process file (LPF); importing, 2–16

Layout

- adding conductors, 7–9
- creating, 2–14
- creating from schematic, 5–1
- exporting, 5–25
- tips and tricks, 5–1
- viewing, 5–9

Layout cell

- anchoring, 5–12
- snapping, 5–21

Layout Manager, 2–4, 2–16

Linear simulators, 4–1

Load line measurement, 6–21

LPF; importing, 2–16

Lumped element filter; creating, 4–1

M

Measurements

- adding, 2–18, 4–8
- Dynamic load line, 6–21
- IP3 vs voltage, 6–25
- Large signal reflection coefficient, 6–13

- nonlinear, 6–1
- Pout vs. frequency, 6–19
- Third-order intermodulation, 6–24

Mesh, 7–22

Microstrip elements; placing, 5–6

MTRACE2

- routing, 5–19

Multi-tone analysis, 6–1

N

Netlists

- creating, 2–4, 2–12

NI AWR Design Environment

- components, 2–3
- design flow, 2–1
- overview, 2–1
- starting, 2–2

Nodes

- connecting, 2–12

Nonlinear

- measurements, 6–1
- model, 6–3
- simulation frequencies, 6–12
- simulator, 6–1

O

Online Help, 1–5, 2–20

Optimization

- goals; adding, 4–14

Optimizing

- circuits, 4–15
- simulations, 2–19

P

Parameter

- editing, 4–6

Plotting

- data files, 3–2
- S-parameter files, 3–2

Ports

- adding, 2–11, 4–5, 7–17
- adding to artwork cell, 5–16
- editing, 2–11
- Two-tone harmonic balance, 6–23

Pout vs. frequency measurement, 6–19

Power amplifier circuit, 6–1

Preview geometry, 7–20

Project

- creating, 2–4, 3–1, 4–1
- default units, 4–1

- examples, 2–5
- opening, 2–4
- saving, 2–4

Project Browser, 2–4

Q

Quick Reference document, 2–2

R

Renormalize

- Smithchart, 3–9

Resources; NI AWR, 1–3

Routing microstrip line, 5–19

S

Schematics

- adding a chip cap cell, 5–17
- adding elements, 4–2
- copying, 6–23
- creating, 3–4, 4–2
- importing, 6–14
- placing data files, 3–4
- simulation, 3–7

Scripts, 2–20

Simulation

- data file directly, 3–2
- EM, 7–1
- frequency, 2–18, 3–6, 4–6, 6–12, 7–18
- nonlinear, 6–1
- optimizing, 2–19
- running, 2–18
- schematic with data file, 3–7
- tuning, 2–19

Single-tone analysis, 6–1

Snapping layout cells, 5–21

Starting the NI AWR Design Environment, 2–2

Status Window, 2–4

Subcircuits

- adding to diagram, 2–11
- adding to schematic, 2–4, 2–11, 6–16
- importing, 2–11

Support

- technical, 1–5
- website, 1–5

Swept variables, 6–25

System diagram

- creating, 2–8

T

Technical Support, 1–5

Third-order intermodulation measurement, 6–24

Tuning

- circuits, 4–11
- simulations, 2–19

Two-tone harmonic balance port, 6–23

V

Variables

- creating, 4–13
- sweeping, 6–25

Vias

- adding, 7–15

Viewing

- layouts, 5–9
- structures in 3D, 7–16

W

Website support, 1–5

Wires

- adding, 4–4

Wizard, 2–20

