

**MWO/VSS 2004**



# ***Getting Started Guide***



**Applied Wave Research, Inc.**

## ***MWO/VSS Getting Started Guide***

Version 6.51, January 2005

**Applied Wave Research, Inc.**  
1960 E. Grand Avenue, Suite 430  
El Segundo, CA 90245

Voice 310.726.3000  
Fax 310.726.3005  
Technical Support [support@appwave.com](mailto:support@appwave.com)  
Website [www.appwave.com](http://www.appwave.com)

© 2005 Applied Wave Research, Inc. All Rights Reserved. Printed in the United States of America. No part of this guide may be reproduced in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the express written permission of Applied Wave Research, Inc.

AWR™, Microwave Office™, Visual System Simulator™, Analog Office™, EMSight™, and AWR Design Environment™ are trademarks of Applied Wave Research, Inc. All other product and company names mentioned herein may be the trademarks or registered trademarks of their respective owners.

The information in this guide is believed to be accurate. However, no responsibility or liability is assumed by Applied Wave Research, Inc. for its use.

# CONTENTS



- 1 INTRODUCING AWR DESIGN ENVIRONMENT ..... 1-1**
  - About This Guide ..... 1-2
  - Getting Additional Information ..... 1-3
  
- 2 INSTALLING MWO/VSS ..... 2-1**
  - Installation Overview ..... 2-1
    - Licensing and Available Features ..... 2-1
    - Installing Downloaded Versions ..... 2-1
    - In Case of Installation Errors..... 2-2
  - Preparing for Installation ..... 2-2
  - Installing the Software ..... 2-3
  - Obtaining Your FLEXlm License ..... 2-4
  
- 3 AWR DESIGN ENVIRONMENT ..... 3-1**
  - Starting AWR Programs ..... 3-2
  - AWR Design Environment Components ..... 3-3
  - Basic Operations ..... 3-5
    - Working With Projects ..... 3-5
    - Working With Schematics and Netlists in MWO ..... 3-6
    - Working with System Diagrams in VSS..... 3-7
    - Using the Element Browser..... 3-8
    - Creating a Layout with MWO ..... 3-13
    - Creating Output Graphs and Measurements ..... 3-16
    - Performing Simulations..... 3-17
    - Using Scripts and Wizards ..... 3-19
  - Using Online Help ..... 3-19

## 4 MWO: USING THE LINEAR SIMULATOR ..... 4-1

Linear Simulations in Microwave Office .....	4-1
Creating a Lumped Element Filter.....	4-1
Creating a New Project.....	4-2
Setting Default Project Units.....	4-2
Creating a Schematic.....	4-3
Creating a Graph .....	4-8
Analyzing the Circuit .....	4-9
Creating Variables.....	4-12
Adding Optimization Goals .....	4-14
Optimizing the Circuit.....	4-14

## 5 MWO: CREATING LAYOUTS FROM SCHEMATICS ..... 5-1

Layouts in Microwave Office .....	5-1
Layout Tips and Tricks .....	5-1
Creating a Layout From a Schematic .....	5-2
Creating a New Project.....	5-2
Importing a Layer Process File .....	5-3
Editing Database Units and Default Grid Size.....	5-3
Importing a GDSII Cell Library.....	5-4
Importing a Data File .....	5-5
Changing the Element Symbol.....	5-6
Placing Microstrip Elements for Layout.....	5-7
Assigning an Artwork Cell to a Schematic Element .....	5-10
Viewing a Layout.....	5-11
Anchoring a Layout Cell.....	5-11
Creating an Artwork Cell .....	5-12
Editing the Schematic and Assigning a Chip Cap Cell.....	5-18
Routing the MTRACE2 Element in Layout.....	5-20
Snapping Functions for Layout Cells .....	5-21
Exporting the Layout.....	5-26

**6 MWO: 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 .....	6-2
Creating a New Project.....	6-2
Creating a Schematic.....	6-3
Creating a Bias Circuit .....	6-6
Importing Schematics .....	6-15
Adding Subcircuits to a Schematic .....	6-16
Creating a Pout vs. Frequency Measurement .....	6-20
Creating a Dynamic Load Line Measurement .....	6-21
Copying a Schematic in the Project Browser .....	6-24

**7 MWO: 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
Creating an EM Structure .....	7-2
Adding Conductors to the Layout .....	7-6
Adding Vias .....	7-9
Viewing the Structure in 3D .....	7-11
Adding Ports and De-embedding Lines .....	7-11
Specifying the Simulation Frequencies.....	7-13
Running the EM Simulator .....	7-14
Displaying Results on a Graph .....	7-16
Changing Frequency Range and Step Size .....	7-17
Animating Currents and Viewing E-Fields .....	7-18
Completing the Filter Layout.....	7-20
Adding a Port .....	7-25
Adding a Measurement to the Graph.....	7-26
Adding an EM Structure to Schematic and Simulating .....	7-28

<b>8 VSS: SYSTEM SIMULATION IN VSS .....</b>	<b>8-1</b>
Overview of VSS Theory.....	8-1
Data Types.....	8-1
Complex Envelope Signal Representation .....	8-2
Center Frequency and Sampling Frequency.....	8-4
Parameter Propagation .....	8-6
AM-Modulation Example.....	8-7
Creating a Project.....	8-7
Setting Default System Settings.....	8-8
Creating a System Diagram.....	8-9
Placing Blocks in a System Diagram .....	8-10
Specifying System Simulator Options .....	8-14
Creating a Graph to View Results .....	8-14
Adding a Measurement.....	8-15
Running the Simulation and Analyzing Results .....	8-17
 <b>9 VSS: END-TO-END SYSTEM .....</b>	 <b>9-1</b>
Creating a QAM Project.....	9-1
Creating a QAM End-to-End System Diagram .....	9-2
Adding Graphs and Measurements .....	9-5
Running the Simulation and Analyzing the Results .....	9-6
Tuning System Parameters.....	9-8
Creating a BER and SER Simulation .....	9-9
Converting BER Curve Results to a Table.....	9-15
 <b>10 VSS: ADDING AN MWO SUBCIRCUIT TO A SYSTEM .....</b>	 <b>10-1</b>
Adding an MWO Filter Circuit to the System.....	10-1
Testing the Filter.....	10-2
Simulating the QAM System.....	10-7
Adding a Graph and Analyzing the Results .....	10-8
Experimenting with Filters .....	10-10
 <b>11 VSS: USING AN MWO NONLINEAR ELEMENT IN VSS ...</b>	 <b>11-1</b>
Importing an Amplifier Model into VSS.....	11-1
Using the Vector Signal Analyzer Block .....	11-4

<b>12 VSS: RF BUDGET ANALYSIS .....</b>	<b>12-1</b>
Creating an RF Chain .....	12-1
<b>13 VSS: VSS EXAMPLES .....</b>	<b>13-1</b>
FSK Example.....	13-1
Using a “Black Box” FSK Modulator .....	13-1
Creating An FSK Modulator Using Elementary Blocks	13-4
Receiver and Demodulation.....	13-7
Adding Graphs and Analyzing Results .....	13-9
I/Q Imbalance Example.....	13-13
I/Q Imbalance and Image Rejection Ratio .....	13-14
End-to-End 64QAM Example .....	13-20
Building a QAM System .....	13-21
Adding Graphs and Analyzing Results .....	13-23
Adjacent-Channel Power Ratio (ACPR) .....	13-29
Mixer Modeling Example.....	13-32
Setting up the System.....	13-32
Adding Graphs and Analyzing Results .....	13-35





# INTRODUCING AWR DESIGN ENVIRONMENT

Welcome to the AWR Design Environment!

The AWR Design Environment comprises two powerful tools that can be used together to create an integrated system and RF design environment: Visual System Simulator™ (VSS) and Microwave Office (MWO). These powerful tools are fully integrated in the AWR Design Environment and allow you to incorporate circuit designs into system designs without leaving the AWR Design Environment.

Microwave Office enables 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 one of Microwave Office's simulation engines -- a linear simulator, an advanced harmonic balance simulator, a 3D-planar EM simulator (EMSight™), 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.

VSS 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'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 provides a real-time tuner that allows you to tune the designs and then see your changes immediately in the data display.

## OBJECT ORIENTED TECHNOLOGY

At the core of MWO and VSS 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 get you up and running quickly in the AWR Design Environment and to show you Microwave Office and VSS 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), or you can download it from the AWR website at [www.appwave.com](http://www.appwave.com).

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

### CONTENTS OF THIS GUIDE

Chapter 2 provides the basic installation procedures.

Chapter 3 provides an overview of the AWR Design Environment including the basic menus, windows, components and commands.

Chapters 4, 5, 6, and 7 take you through hands-on examples that show you how to use Microwave Office to create circuit designs.

Chapters 8, 9, 10, 11, and 12 take you through hands-on examples that show you how to use VSS to create system simulations and to incorporate Microwave Office circuit designs.

### CONVENTIONS USED IN THIS GUIDE

This guide uses the following typographical conventions:

Item	Convention
Anything that you select (or click) in the AWR Design Environment, like menu items, button names, and dialog box option names	Shown in a bold 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 <b>File &gt; New Project</b>
Any text that you enter using the keyboard	Shown in a bold type within quotation marks:  Enter " <b>my_project</b> " in <b>Project Name</b> .

Item	Convention
Keys or key combinations that you press	Shown in a bold 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 <b>Alt+F1</b> .

## GETTING ADDITIONAL INFORMATION

There are multiple resources available for additional information and technical support on MWO and VSS.

### DOCUMENTATION

Documentation for MWO includes:

- *What's New in MWO/AO 2004?* presents the new or enhanced features, elements, and measurements for this release. This document is available via the **Start** button **Programs > AWR Suite 2004** menu, or by choosing **Help > What's New** while in the program.
- *Microwave Office/Analog Office User Guide* describes how to use the Microwave Office windows, menu choices, and dialog boxes to perform linear, nonlinear, and EM design, layout, and simulation, and discusses related concepts.
- *Microwave Office/Analog Office Element Catalog (Volumes 1 and 2)* provides complete reference information on all of the electrical elements that you use to build schematics.
- *Microwave Office/Analog Office Measurement Reference* provides complete reference information on the "measurements" (i.e., computed data such as gain, noise, power, or voltage) that you can choose as output for your simulations.
- *MWO/VSS/AO Installation Guide* describes how to install the AWR Design Environment 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 Applied Wave Research website at [www.appwave.com](http://www.appwave.com) under Support.

- *Known Issues* lists the known issues for this release. This document is available on your program disk as *KnownIssues.htm*.

Documentation for VSS includes:

- *What's New in VSS 2004?* presents the new or enhanced features, system blocks, and measurements for this release. This document is available via the **Start** button **Programs > AWR Suite 2004** menu, or by choosing **Help > What's New** while in the program.
- *Visual System Simulator System Block Catalog* provides complete reference information on all of the system blocks that you use to build systems.
- *Visual System Simulator Measurement Reference* provides complete reference information on the measurements you can choose as output for your simulations.
- *VSS Modeling Guide* contains information on simulation basics, RF modeling capabilities, and noise modeling.

## ON-LINE HELP

All AWR documentation is available as on-line Help.

To access online Help choose **Help** from the main menu or press **F1** anytime you are working within the AWR Design Environment. 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 Measurements dialog box.

## WEBSITE SUPPORT

Support is also available from the Applied Wave Research website at [www.appnwave.com](http://www.appnwave.com). You can go directly to this site from the AWR Design Environment **Help** menu. The Support page provides links to the following:

- the current software version
- the KnowledgeBase, which contains Frequently Asked Questions (FAQs) from MWO and VSS users, Application Notes, Tutorials, and project examples
- VSS and MWO documentation

## TECHNICAL SUPPORT

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

Phone: 310.726.3028 / Fax: 310.726.3005 / E-mail: [support@appwave.com](mailto:support@appwave.com).  
For users with current maintenance contracts, Technical Support is also available via an interactive Web-based chat session. To contact Support in this manner, choose **Help > Chat with AWR Support**.



# INSTALLING MWO/VSS

---

This chapter describes how to install Microwave Office (MWO) and Visual System Simulator (VSS). You can install them as standalone applications or install them together as integrated partners within the AWR Design Environment. A procedure for obtaining a FLEXlm<sup>®</sup> license with a software-based key is also included.

The installation procedures are intended for evaluators and licensed users who want to install MWO and VSS with a FLEXlm license dedicated to their particular machine. For alternative licensing configurations, see the *MWO/VSS/AO Installation Guide* on your Program Disk (*install.pdf*). You can also download this guide from the AWR website at [www.appnwave.com](http://www.appnwave.com). The file is located under Support.

## INSTALLATION OVERVIEW

---

The AWR Design Environment software is shipped on a program CD-ROM for installation. The installation program installs Microwave Office, Visual System Simulator and Analog Office.

### Licensing and Available Features

You may have purchased a complete MWO/VSS license with full functionality (i.e., linear simulator, nonlinear simulator, EM simulator, optional HSPICE simulator, and layout tool), or you may have purchased a license for one or more features. In either case, the complete application is installed and your license determines the specific MWO and/or VSS functions that are available to you. The program is installed by default in *C:\Program Files\AWR\AWR2004*.

### Installing Downloaded Versions

You can also download and install the program from the Applied Wave Research website ([www.appnwave.com](http://www.appnwave.com)). You can access XML libraries from within the AWR Design Environment via the Element Browser, and on the AWR website.

## In Case of Installation Errors

If installation aborts due to an error, ensure the machine's TEMP environment variable points to an existing directory. To set environment variables, see the *MWO/VSS/AO Installation Guide* (*install.pdf* on your Program Disk, or under Support at [www.appnwave.com](http://www.appnwave.com).)

## PREPARING FOR INSTALLATION

---

Before you start the installation:

- 1 Make sure the machine on which you wish to install the software meets the following hardware and software requirements:
  - Pentium<sup>®</sup> PC, 800MHz, 256MB RAM, 200MB available disk space.
  - Microsoft<sup>®</sup> Windows<sup>®</sup> 2000 or XP.
  - Microsoft Internet Explorer Version 5.0 or later, including Web Browser<sup>1</sup>, Help, and Visual Basic scripting support. To install Explorer, go to the Microsoft website at <http://www.microsoft.com/windows/IE>, or insert the Program Disk into the CD-ROM drive and run *D:\Tools\ie6.sp1\i386\ie6.setup.exe*, where *D:* is the CD-ROM drive designation.
- 2 When installing an update, back up any custom libraries you have stored in the *\Library* subdirectory of the program directory, and any *.dll* files you have stored in the *\cells* or *\models* directories.
- 3 When installing an upgrade, retain your existing version until you verify that your projects work successfully in the new version. (To uninstall Microwave Office, choose **Add/Remove Programs** from the Windows Control Panel, select the AWR Design Environment and follow the instructions.)

---

1. If you do not want Microsoft Internet Explorer to be your default internet browser, you must choose to NOT associate file types via the Advanced setup options when you install Internet Explorer. Note that you must still have the Web Browser installed, as the AWR Help and XML libraries use it.



## INSTALLING THE SOFTWARE

You can begin your installation from the installation Welcome screen.

To install the AWR Design Environment:

- 1 If you have the program CD-ROM, place it into the CD-ROM drive. When the AWR splash screen displays, follow the prompts and instructions. If the Welcome dialog does not display, explore the program CD-ROM and run the *setup.exe* program.

If you have downloaded MWO and VSS from [www.awpware.com](http://www.awpware.com), browse to the folder in which you downloaded the software and run the *setup* program to display the installation Welcome dialog.

- 2 As you proceed with the installation you are prompted to specify the following:

Option	Description
Select Destination Directory	Select the directory in which you want to install the AWR Design Environment files.
Select Default Process	Choose the default units to use in schematics and layouts (as well as affect the default sizes for components such as transmission lines). The default is Microns. You can alternatively set this default within the program; see the <i>Microwave Office User Guide</i> for details.
Register File Extensions	Select this check box to specify that files with a <i>.emp</i> , <i>.em</i> , <i>.sch</i> , or <i>.net</i> extension are opened in the AWR Design Environment. If you use another schematic tool or program that uses these extensions, you may want to disable this option. This check box is selected by default.
Select Start Menu Group	Enter the name of the <b>Start</b> menu program group to which you want to add the AWR 2004 icons. "AWR Suite 2004" is the default. If you accept the default, you start the program by clicking the <b>Start</b> button on your desktop and then choosing <b>Programs &gt; AWR Suite 2004 &gt; AWR Design Environment</b> . In addition, you can specify that these icons display only when you are logged onto the computer, or when anyone logs onto the computer.

- 3 When prompted to create one or more **Start** menu shortcuts, choose from the following list of AWR product numbers based on the product feature set you have purchased.

Option	Description
Evaluation	Select this option to create the appropriate default shortcut.

Option	Description
MWO-328	All MWO-228 functionality plus HSPICE functionality
MWO-228	Linear, nonlinear, and EM simulators, and layout; design rule checking, ability to work with layout components having more than two layers, and access to foundry libraries
MWO-225	linear, nonlinear, and EM simulators, and layout
VSS-100	Standalone version of Visual System Simulator
MWO-205	linear and nonlinear simulators, and layout
MWO-200	linear and nonlinear simulators
MWO-125	linear and EM simulators, and layout
MWO-120	linear and EM simulators
MWO-105	linear simulator and layout
MWO-100	linear simulator

- 4 When installation is complete click **Finish** to close the screen.

## OBTAINING YOUR FLEXLM LICENSE

The AWR Design Environment supports FLEXlm floating<sup>1</sup> and locked licensing as well as hardware-based<sup>2</sup> keys. If you want to configure your site for FLEXlm floating licensing, and/or if you want to use hardware-based keys, see the *MWO/VSS/ANO Installation Guide* under Support at [www.appware.com](http://www.appware.com), or on your Program Disk (*install.pdf*).

To obtain a FLEXlm locked license using a software-based key:

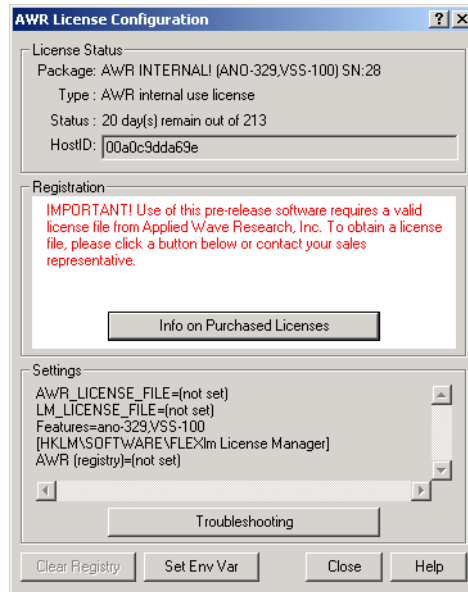
- 1 Click the **Start** button on your desktop, and choose **Programs > AWR Suite 2004 > AWR Design Environment**.

If the AWR Design Environment was not configured during installation to display from your **Start** menu, start it by double-clicking **My Computer** on

- 
1. Floating licensing allows multiple users to share a license over a network via a client-server architecture, whereas locked licensing dedicates a license to a particular machine.
  2. Hardware-based keys are calculated from an AWR-supplied hardware dongle serial number. Such a license can be transferred between machines simply by moving the dongle.

your desktop, opening the folder where you installed the program and double-clicking on *MWOOffice.exe*.

- 2 The following AWR License Configuration dialog box displays.

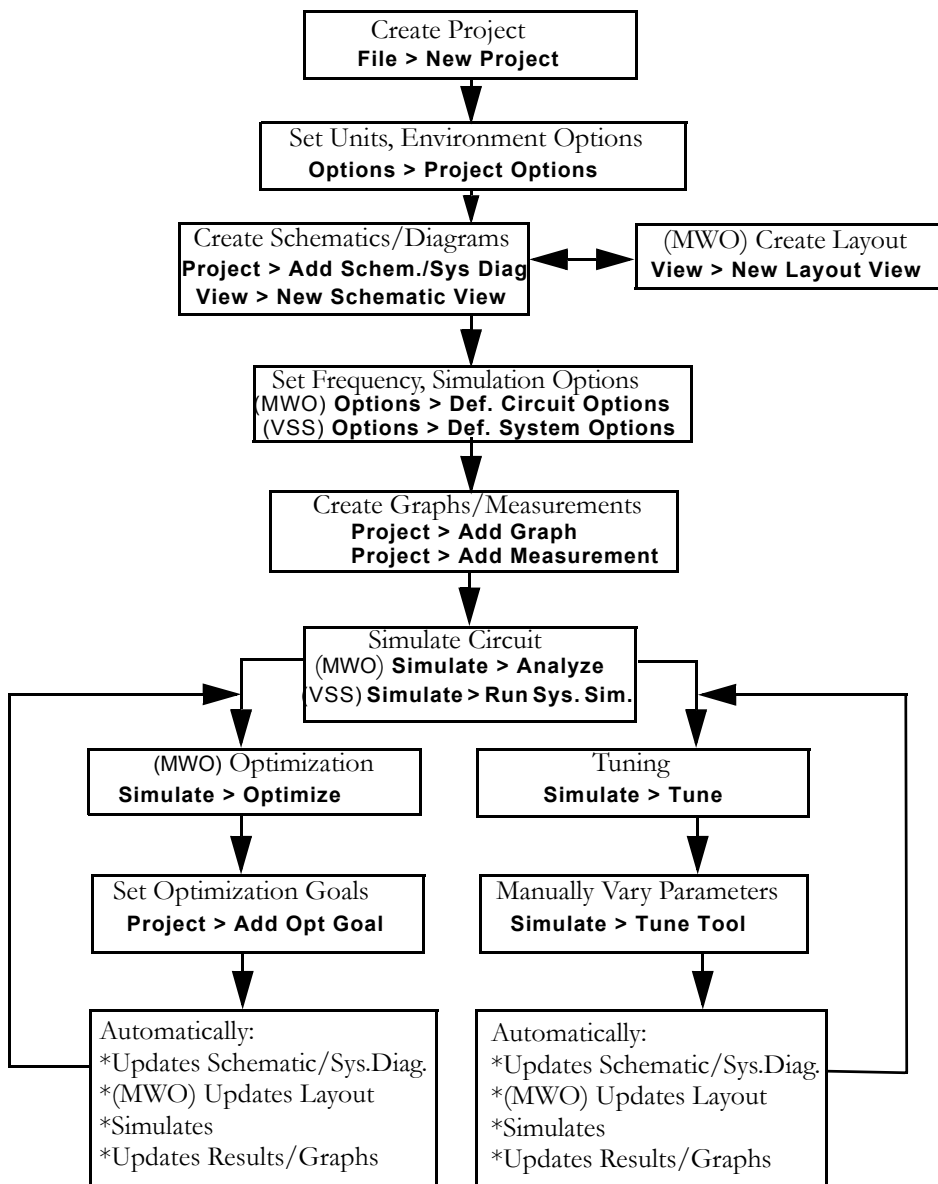


- 3 To obtain a valid license file from Applied Wave Research, click **Info on Purchased Licenses** and follow the instructions.
- 4 You will receive your license within two business days. When you receive it, rename it *awr.lic* and place it into the program directory.



# AWR DESIGN ENVIRONMENT

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



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

- 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

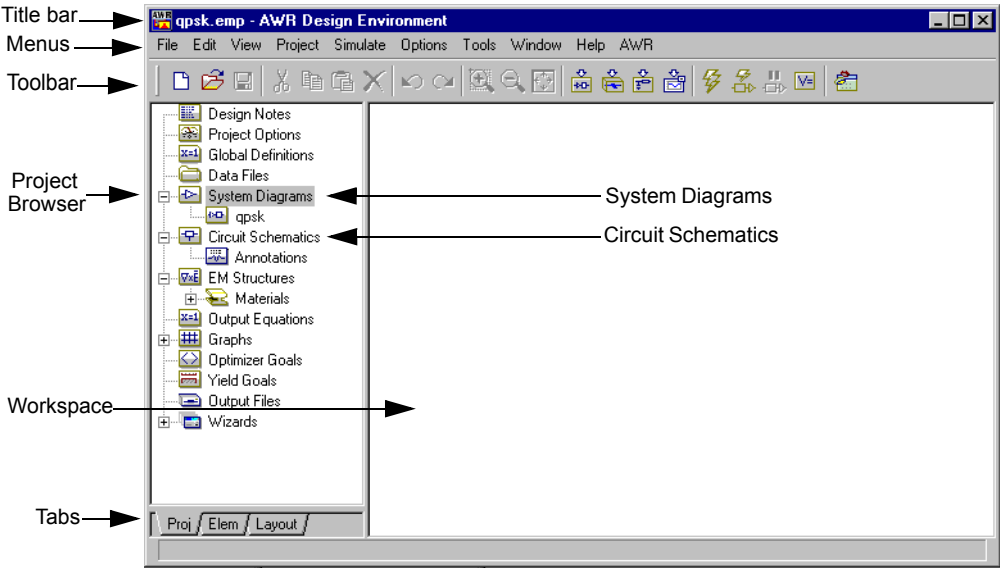
## STARTING AWR PROGRAMS

---

To start the AWR Design Environment:

- 1 Click **Start** on your desktop.
- 2 Choose **Programs > AWR Suite 2004 > AWR Design Environment**. The following AWR Design Suite main window displays.

If the AWR Design Environment 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 **MWOffice.exe**, the Microwave Office application.



## AWR DESIGN ENVIRONMENT COMPONENTS

The AWR Design Environment 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 both Microwave Office (MWO) and Visual System Simulator (VSS). The major components of the AWR Design Environment are:

Component	Description
menu	A set of menus located along the top of the window for performing a variety of MWO and VSS tasks.
toolbar	<p>A row of buttons located just below the menu that provides shortcuts to frequently used commands such as creating new schematics, performing simulations, or tuning parameter values or variables.</p> <p>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 a short description of the button.</p>

Component	Description
Project Browser	<p>Located 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, and more.</p> <p>The Project Browser is active when the AWR Environment first opens, or when you click the <b>Proj</b> tab at the lower left of the window. Right-click a node in the Project Browser to access menus of relevant commands.</p>
Element Browser	<p>Contains a comprehensive inventory of circuit elements for building your schematics, and system blocks for building system diagrams for simulations. The Element Browser displays in the left column in place of the Project Browser when you click the <b>Elem</b> tab at the lower left of the window.</p>
workspace	<p>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 <b>View</b> menu.</p>
tabs	<p>A set of tabs located at the lower left of the window that allow you to switch the contents of the left column of the window from Project Browser (the <b>Proj</b> tab) to Element Browser or Layout Manager.</p> <p>Click the <b>Elem</b> tab to display the Element Browser and to access a comprehensive inventory of circuit elements and system blocks for simulation.</p> <p>Click the <b>Layout</b> tab to specify options for viewing and drawing layout representations and to create new layout cells.</p>

You can invoke many of the functions and commands from the menus and on the toolbar, and in some cases by right-clicking on 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 AWR Design Environment.

### 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 MWO and VSS are fully integrated in the AWR Environment, you can start a project based on a system design using VSS, or on a circuit design using MWO. The project may ultimately combine both VSS and MWO 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.

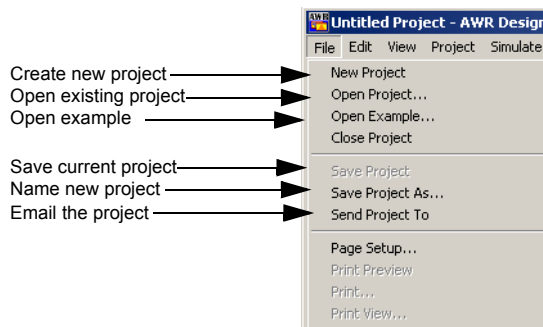
#### CREATING, OPENING, AND SAVING PROJECTS

When you first start the AWR Design Environment, 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.

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. AWR projects are saved as \*.emp files.

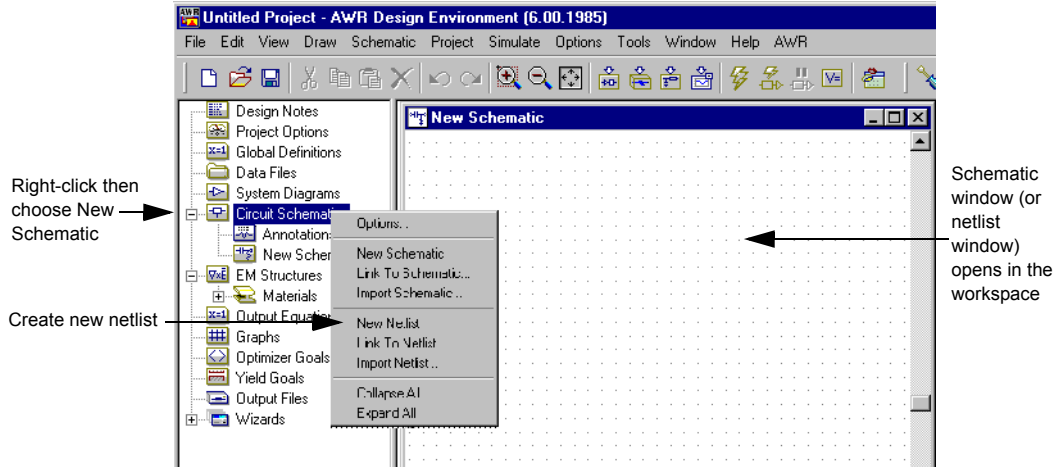


## Working With Schematics and Netlists in MWO

A schematic is a graphical representation of a circuit while a netlist is a text-based description. A Microwave Office project can include multiple linear and nonlinear schematics and netlists.

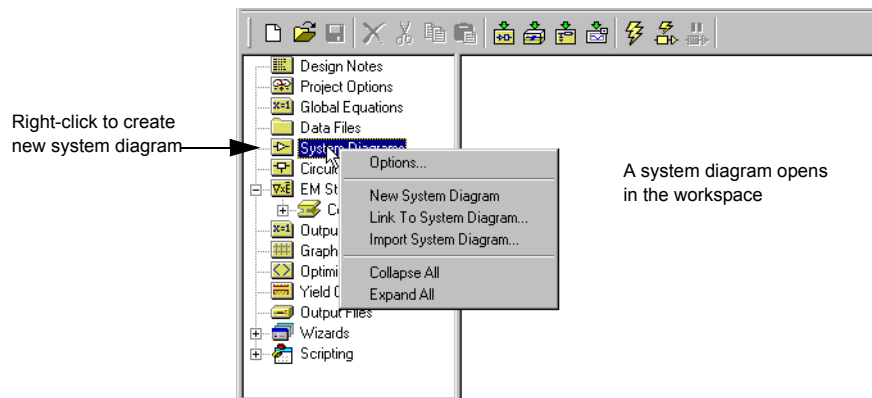
To create a schematic or netlist, right-click **Circuit Schematics** in the Project Browser and choose **New Schematic** or **New Netlist**.

After you specify a schematic or netlist name, a schematic or netlist window opens in the workspace and the Project Browser displays the new item as a subnode under **Circuit Schematics**. Subnodes of the new schematic or netlist which contain all of the parameters and options (for example, frequencies or harmonic balance options) that define and describe the schematic or netlist can be displayed in the Project Browser by selecting **Options > Environment Options > Show options as sub-nodes**. In addition, the menu and toolbar display new choices particular to building and simulating schematics or netlists.



## Working with System Diagrams in VSS

A VSS project can include multiple system diagrams. To create a system diagram, right-click **System Diagrams** in the Project Browser and choose **New System Diagram**.



You are prompted to name the new system diagram.

After you name the system diagram, a window opens in the workspace and the Project Browser displays the new system diagram. Subnodes of the system diagram which contain all of the parameters and options that define and describe the system diagram can also be displayed in the Project Browser by selecting **Options > Environment Options > Show options as sub-nodes**. After

you name the system diagram, the menus and toolbar display new selections and buttons for building and simulating systems.

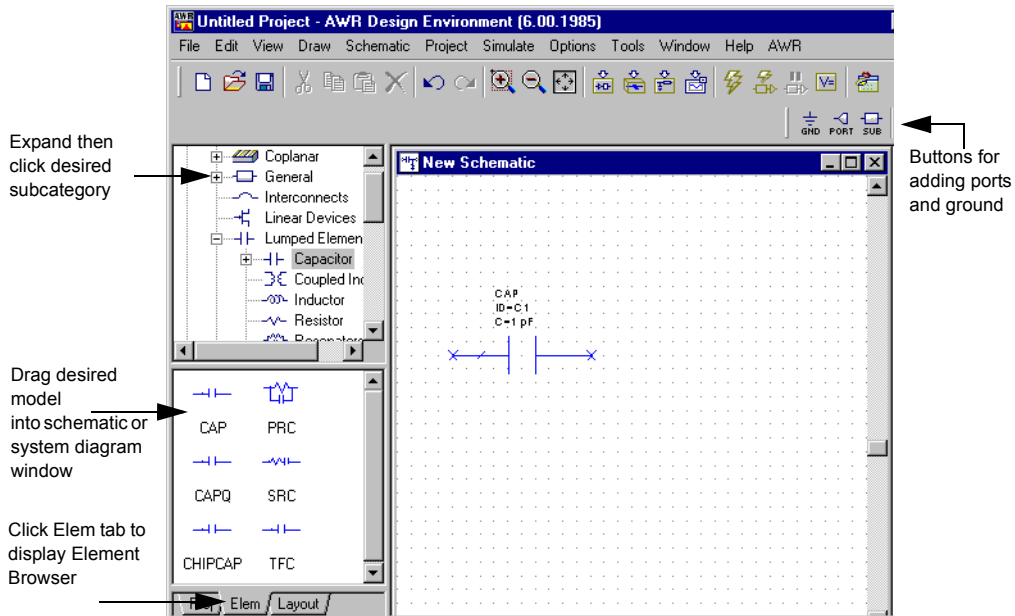
## Using the Element Browser

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 XML Libraries folders in the Element Browser provide 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 **Elem** tab in the lower left window. The Element Browser replaces the Project Browser window.
- To expand and contract 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 on the value in the schematic or system diagram and entering a new value in the text box that displays.



## ADDING SUBCIRCUITS TO SCHEMATICS

Subcircuits allow you to construct hierarchical circuits by including a circuit block in a 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 add a data file as a subcircuit, you must first import it and add it to the project as a node. To do so, choose **Project > Add Data File**. Any imported data files are automatically shown 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 AND WIRES TO SCHEMATICS AND 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 **Port** or **Ground** buttons on the toolbar, position the port or ground, and click to place it.

### 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.



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. To

start a wire from another wire, **Shift**-click to mark the beginning of the wire. Terminate the wire by clicking on another element node or on top of another wire. To cancel a wire, press the **Esc** key.

## EDITING PORT PARAMETERS

To edit port parameters, double-click the port in the schematic or system diagram windows to display a dialog box in which you can specify new parameter values.

## ADDING DATA TO NETLISTS

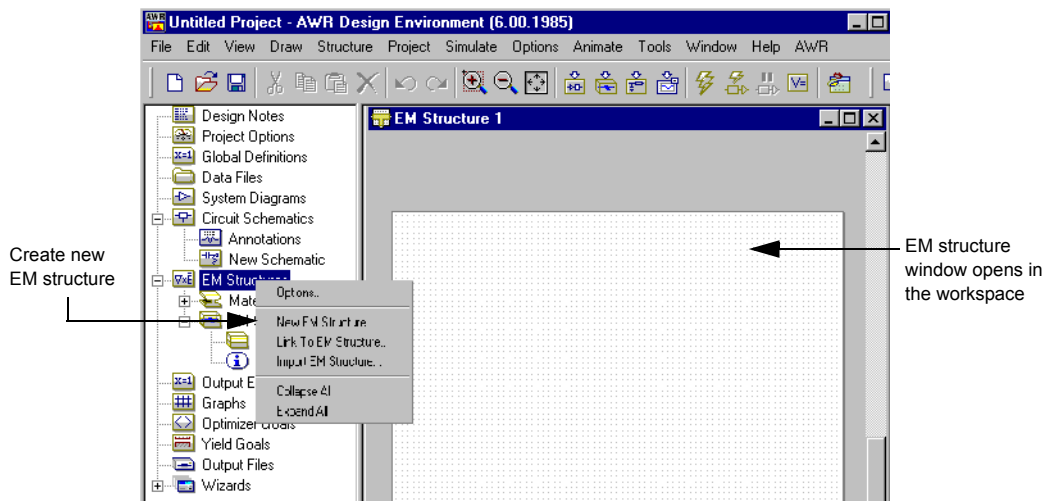
When you create a netlist, an empty netlist window opens into which you type a text-based description of a schematic. Netlist data is arranged in blocks in a particular order, where each block defines a different attribute of an element such as units, equations, or element connections. For more information about creating netlists, see the *Microwave Office 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 **EM Structures** 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 all of the parameters and options that define and describe the EM structure can be displayed as described in “Working With Schematics and Netlists in MWO” on page 3-6. In addition, the menu and toolbar display new choices particular to drawing and simulating EM structures.



## ADDING EM STRUCTURE DRAWINGS

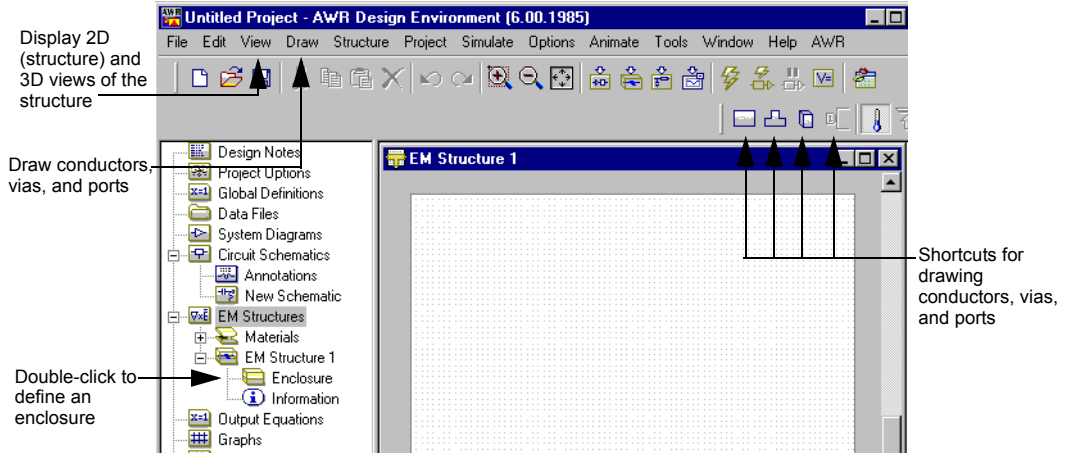
Before you draw an EM structure, you must define an enclosure. The enclosure specifies things such as boundary conditions and dielectric materials for each layer of the structure.

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 create drawings by accessing options from the **Draw** menu to draw components such as rectangular conductors, vias, and edge ports.

You can view EM structures in 2D (structure) and 3D by using the **View** menu, and you can view currents and electrical fields using the **Animate** menu.





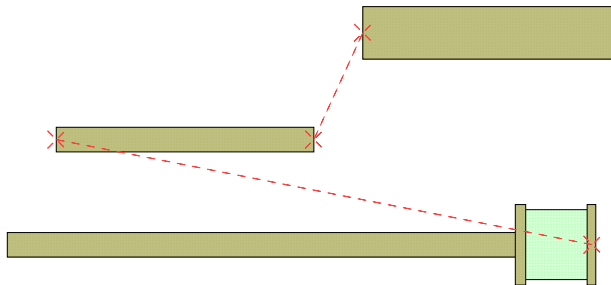
## Creating a Layout with MWO

A layout is a view of the physical representation of a circuit, in which each component of the schematic is assigned a layout cell. In the object-oriented AWR environment, 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.

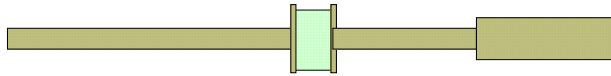
To create a layout representation of a schematic, click the schematic window to make it active, and choose **View > New Layout View**. A layout window displays (overlying the schematic window) with an automatically-generated layout view of the schematic.



You can also click the **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.

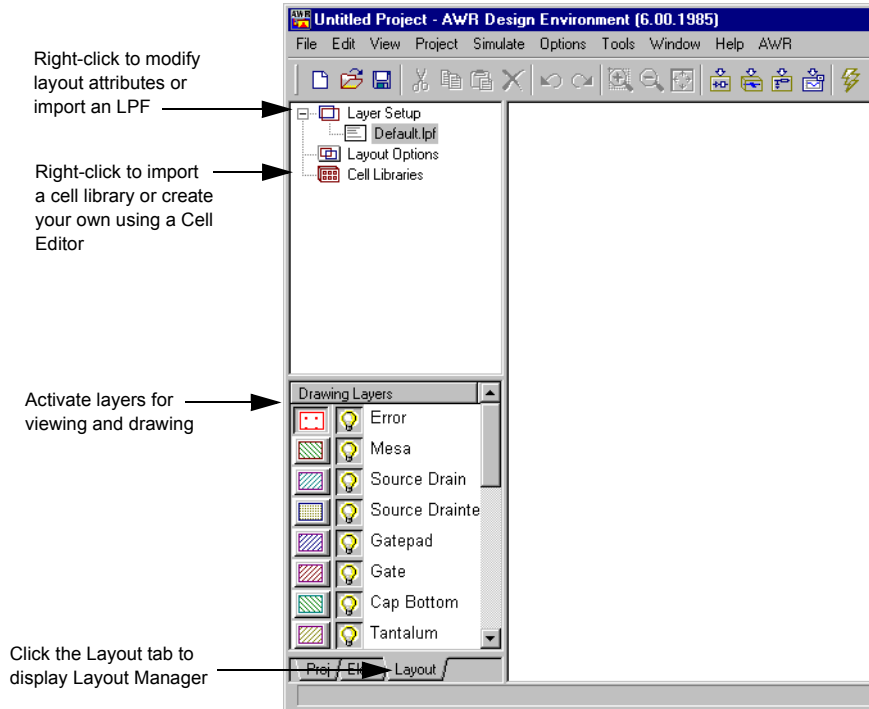


When you choose **View > New Layout View**, default layout cells are automatically assigned for common electrical components such as microstrip, coplanar waveguide, and stripline elements. Components of the schematic that do not map to default layout cells display in blue in the schematic window after the layout is generated; components that do have default layout cells display in magenta. For components without default layout cells defined, you must create them or import them using the Layout Manager. For more information see *Using the Layout Manager* on page 3-15.

You can draw in the layout window using the draw tools to build substrate outlines, draw DC pads for biasing, or to add other elements.

### MODIFYING LAYOUT ATTRIBUTES AND DRAWING PROPERTIES

To modify layout attributes and drawing properties, as well as create new layout cells for elements that do not have default cells, click the **Layout** tab in the lower left window. The Layout Manager replaces the Project Browser window.



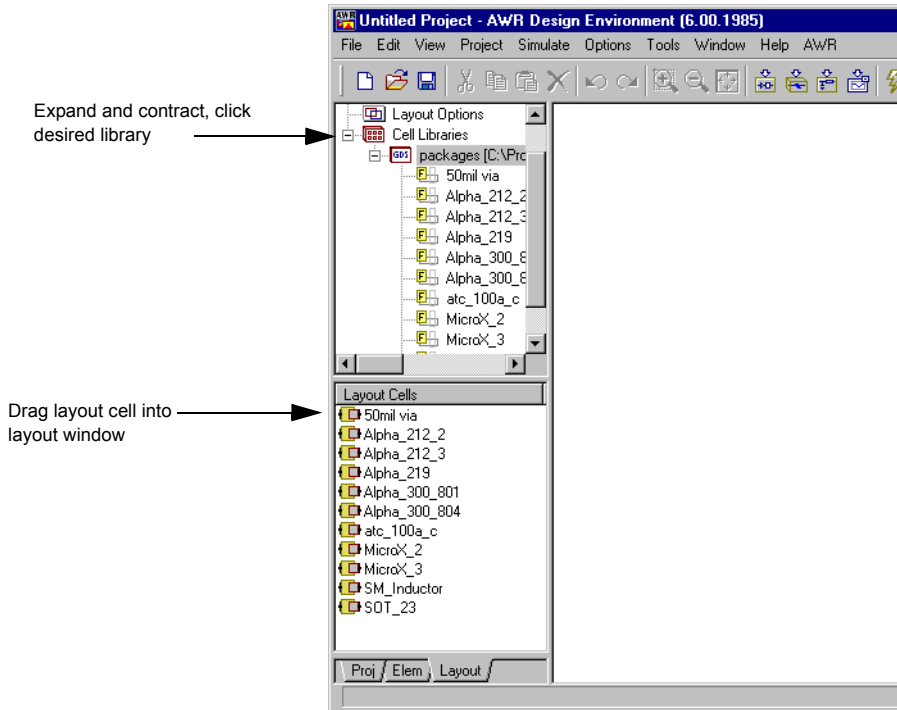
## USING THE LAYOUT MANAGER

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, right-click **Layer Setup** and choose **Edit Drawing Layers**. 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 coordinate entry, boolean operations for subtracting and uniting shapes, array copy, arbitrary rotation, grouping, and alignment tools. You can also import artwork cell libraries such as GDSII or DXF into the AWR Design Environment.

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. To place a cell into the layout window, simply click and drag the cell, release the mouse button, position it, and click to place it.



You can import layouts as GDSII or DXF files. To export a layout, click the layout window to make it active, and choose **Layout > Export**.

## 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 create a graph and specify the data, or measurements, that you want to plot. Measurements can include for example, gain, noise or scattering coefficients.

To create a graph, right-click **Graphs** in the Project Browser, and choose **Add 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.
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 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 allows you to choose from a comprehensive list of measurements.

To compare the existing graphs with different simulation settings, while the graph window is active click **Graph** on the menu bar, choose **Freeze Traces**, then make the necessary changes and run the simulation again.

## Performing Simulations



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 pieces of the project.

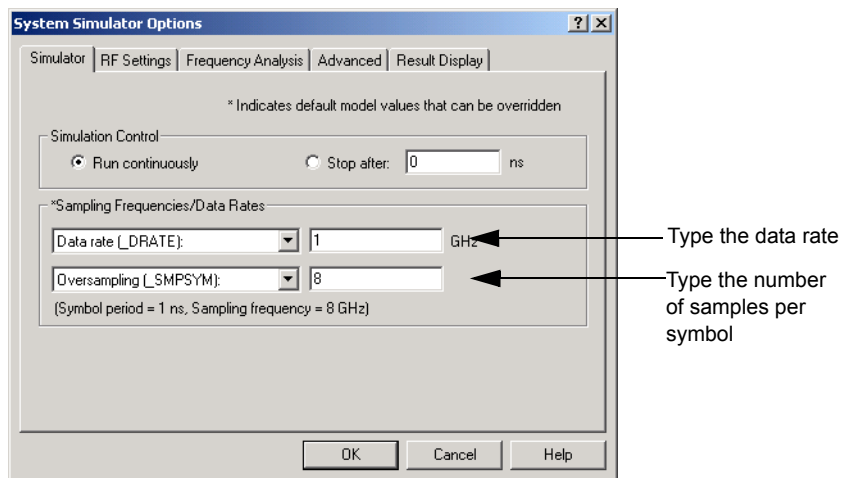
### SETTING SIMULATION FREQUENCY

To set the 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 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, deselect **Use project defaults** and then specify frequency values. When the simulation is complete, you can view its output on the graphs and then easily tune and/or optimize as needed.

## VSS SYSTEM SIMULATIONS

To set 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 **Simulator** tab in the dialog box.



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

## 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.



You can also click the **Tune Tool** button on the toolbar. Select the parameters you want to tune and then click the **Tuner** 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 Scripts and Wizards

AWR Design Environment scripts and wizards allow you to automate and extend Microwave Office functions in a non-proprietary manner. 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 Basic programs that you can write to do things such as automate schematic-building tasks within Microwave Office.

Wizards are Dynamic Link Library (DLL) files which you can author to create add-on tools for Microwave Office, for example, a filter synthesis tool, and Load Pull tool.

Wizards display under **Wizards** in the Project Browser; to access scripts, choose **Tools > Scripting Editor**.

## USING ONLINE HELP

---

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

To access Help, choose **Help** from the menu bar or press **F1** anywhere in the program. The following context-sensitive Help is also available:

- Help buttons in most dialog boxes.
- Help for each element or system block in the Element Browser by selecting a model and pressing **Alt+F1**, or by right-clicking on a model and choosing **Element Help**. The Element Options dialog box also has an **Element Help** button.
- Help for each measurement in the Add/Modify Measurement dialog box by clicking **Meas Help**.
- Help for using the AWR script development environment, accessed by selecting a keyword (i.e., object, object model, or Visual Basic syntax), and pressing **F1**.





# 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 is architected using object-oriented techniques that enable fast and efficient simulations of linear circuits. One of its trademarks is a real-time tuner that allows you to see resulting simulations as you tune. It also allows you to perform 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 to simulate a basic lumped element filter using the linear simulator. It includes the following steps:

- Creating a schematic
- Adding graphs and measurements
- Analyzing the circuit
- Tuning the circuit
- Creating variables
- Optimizing the circuit

## Creating a New Project

The example you create in this chapter is available in its complete form as “linear\_example.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\Microwave Office\Linear* directory. You can use this example file as a reference.

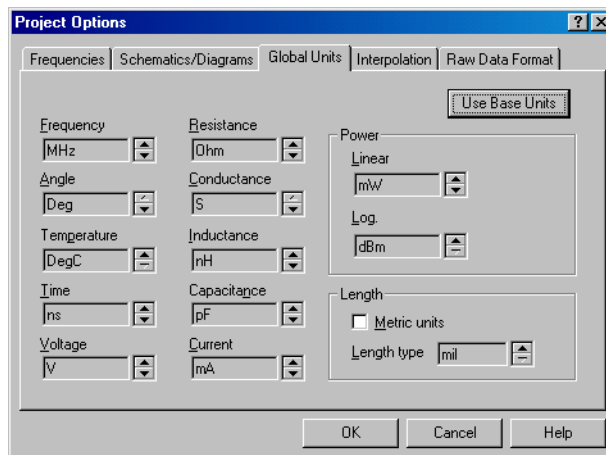
To create a project:

- 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**.

## Creating a Schematic

To create a schematic:

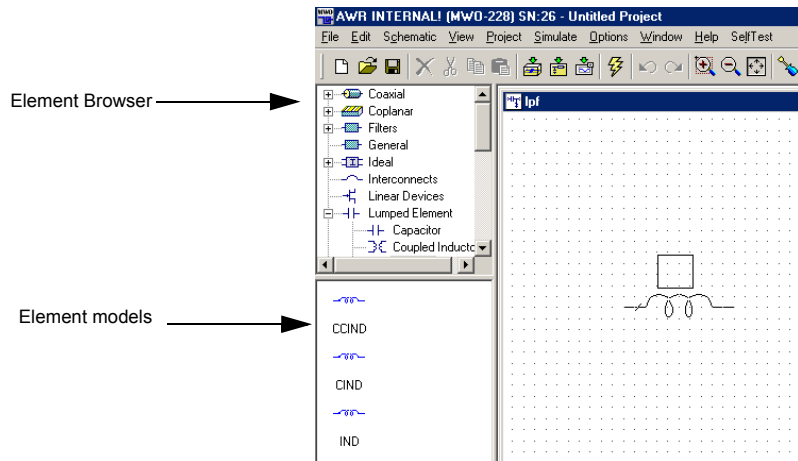
- 1 Choose **Project > Add Schematic > New Schematic**. The Create New Schematic dialog box displays.
- 2 Type “**lpf**”, and click **OK**. A schematic window displays in the workspace and the schematic displays under **Circuit Schematics** in the Project Browser.

### 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 **Elem** tab in the lower left of the window 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 Expand the **Lumped Element** category under **Circuit Elements**, then click the **Inductor** subgroup. Select the IND model from the bottom window and drag it to the schematic as shown in the following figure.



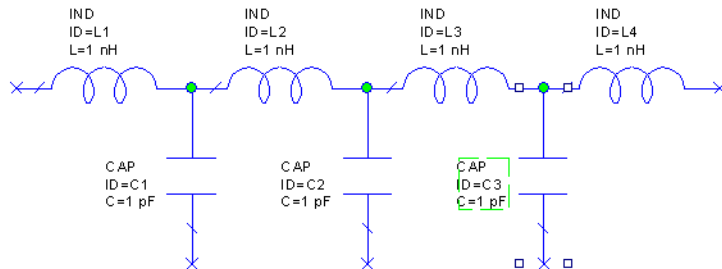
- 4 Repeat step 3 three times, aligning and connecting each inductor as shown in the following figure.

**TIP:**

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

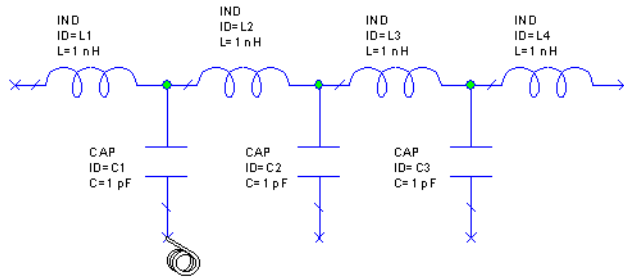


- 5 Click the **Capacitor** subgroup under **Lumped Element**, then select the CAP model and place it on the schematic as shown in the following figure. Right-click once before placing the capacitor to rotate it as shown.
- 6 Repeat Step 5 twice, aligning and linking each capacitor as shown in the following figure.

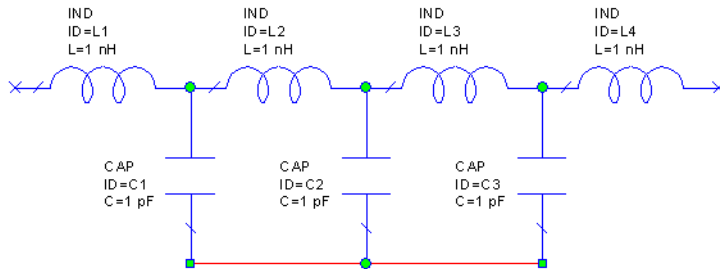
**CONNECTING THE WIRES**

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 as a wire coil symbol as shown in the following figure.



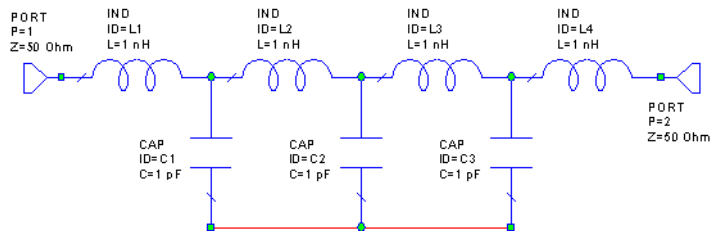
- 2 Click, then drag the wire past the bottom node of CAP C2, then onto the bottom node of CAP C3, and click to place the wire.



## PLACING PORTS ON A NODE

To place a port on a node:

- 1 Choose **Draw > Add Port**.
- 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.



- Repeat Step 1 to add a port to the right-most inductor, but right-click two times to rotate the port 180-degrees before you place it.

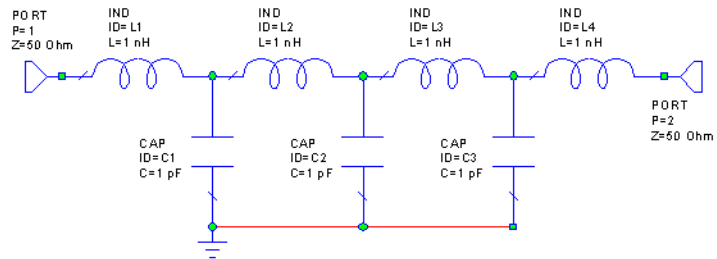
**TIP:**

You can also add a port by clicking the **Port** button on the toolbar and sliding the cursor into the schematic.

**PLACING GROUND ON A NODE**

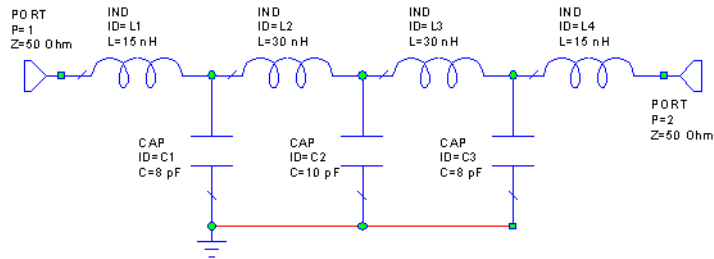
To place ground on a node:

- Choose **Draw > Add Ground**.
- Slide 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.

**EDITING ELEMENT PARAMETERS**

To edit the element parameters:

- Double-click the IND L1 element. The Element Options dialog box displays.
- Set the L parameter Value to “**15**” and click **OK**. The change is reflected in the schematic.
- 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.)

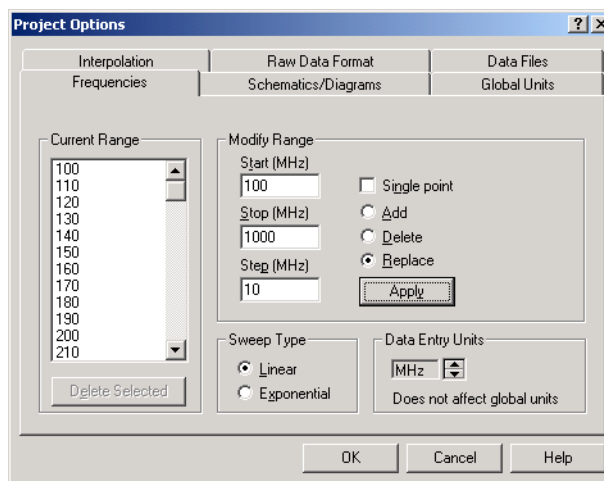


**TIP:** You can also simply double-click the parameter value displayed on the schematic to display a text box in which you can modify a single parameter.

## SPECIFYING THE SIMULATION FREQUENCY

To specify the simulation frequency:

- 1 Click the **Proj** tab in the lower left of the window.
- 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**.



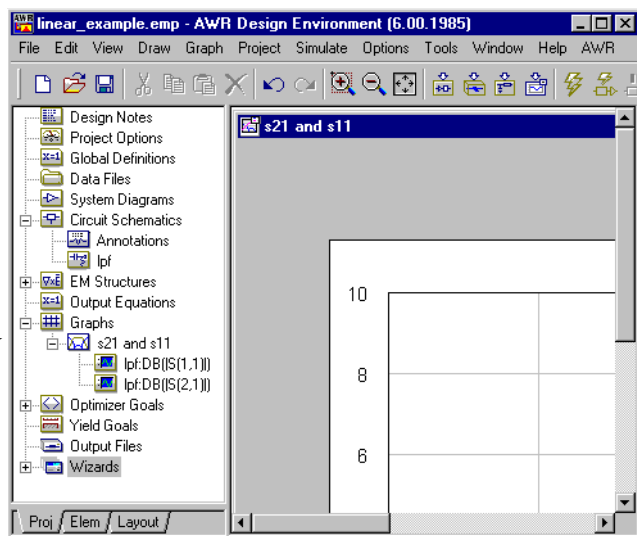
## Creating a Graph



To create a graph:

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

Individual graphs display under **Graphs** →



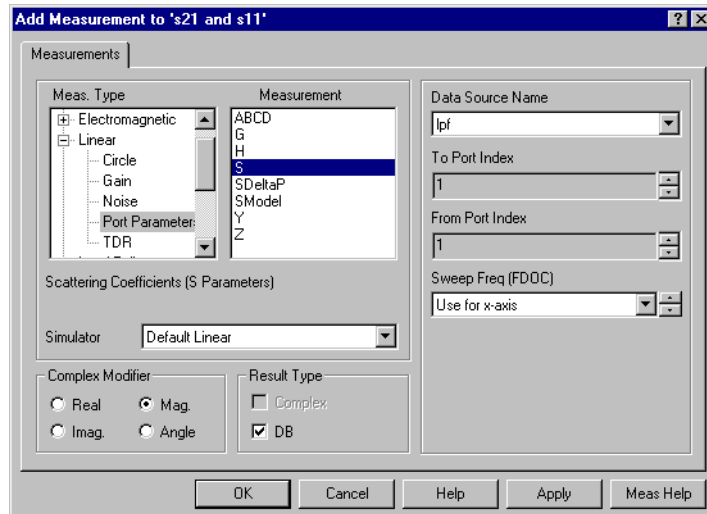
## ADDING A MEASUREMENT

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 Measurement** button on the toolbar.
- 2 Select **Linear > Port Parameters** as the **Meas. Type** and **S** as the **Measurement**. Click the arrow to the right of **Data Source Name** and select **Ip1**. 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 **Apply**.





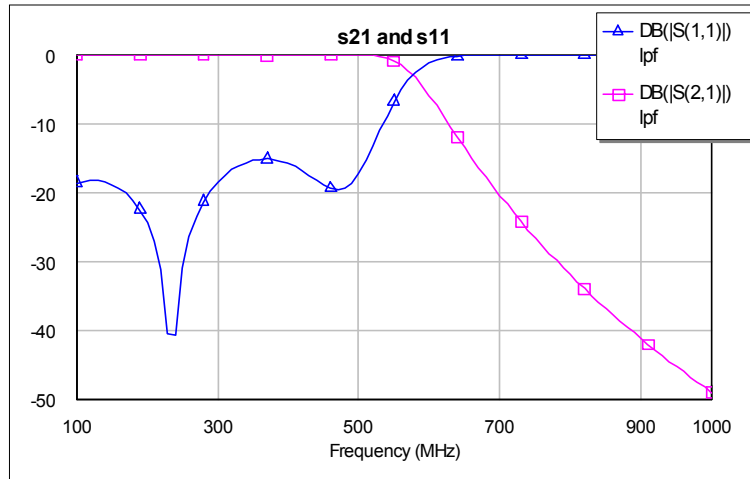
- 3 Change the value in **To Port Index** to “2”, and click **Apply** to add a second measurement.
- 4 Click **OK**. The measurements  $\text{lpf:DB}(|S(1,1)|)$  and  $\text{lpf:DB}(|S(2,1)|)$  display under the “s21 and s11” graph in the Project Browser.

## Analyzing the Circuit

To analyze the circuit:



- 1 Choose **Simulate > Analyze**. The simulation response displays on the graph. You can also click the **Analyze** button on the toolbar to simulate the active project.

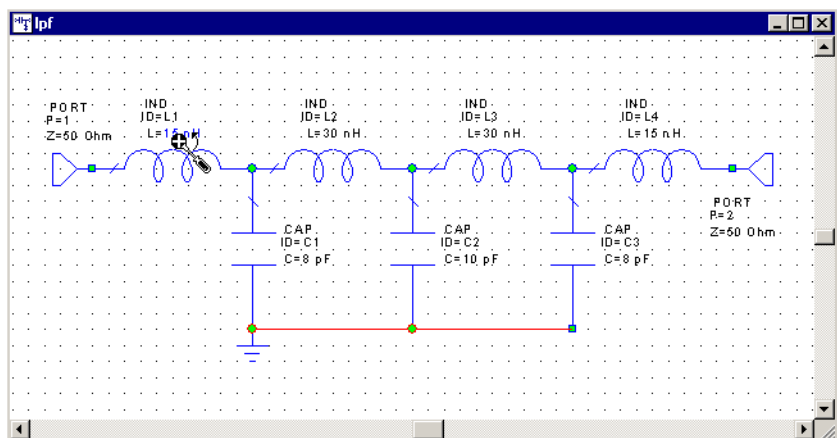


### TUNING THE CIRCUIT

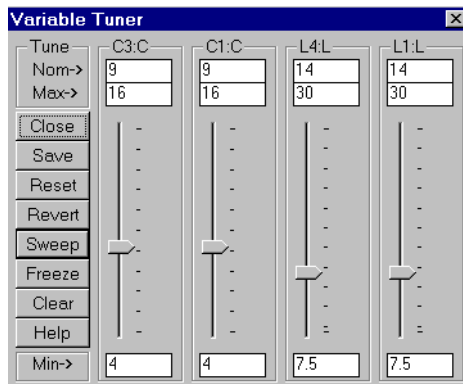
When you place the tune tool over a schematic element, the cursor displays as a cross icon to indicate that the parameter is tuneable.

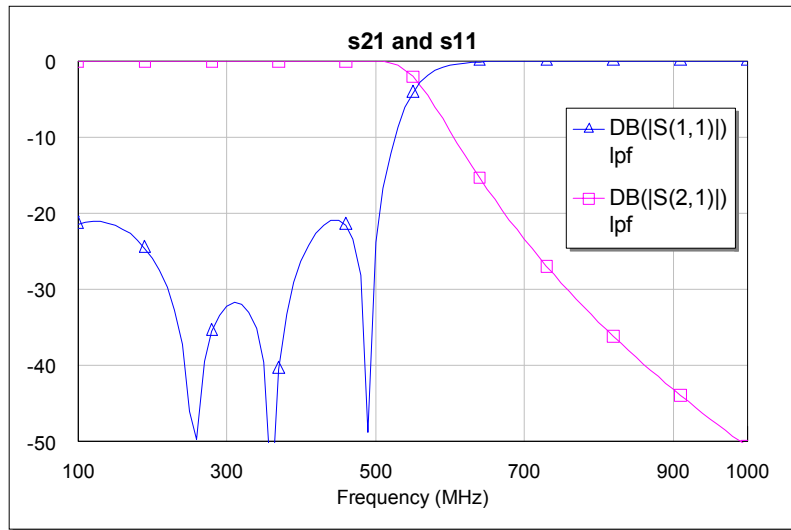
To tune the circuit:

- 1 Click the schematic window to make it active.
- 2 Click the **Tune Tool** button on the toolbar.
- 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 Repeat Steps 2 through 4 for the IND L4 element and the C parameters of the CAP C1 and CAP C3 elements.
- 6 Click the graph window to make it active.
- 7 Choose **Simulate > Tune**. The Variable Tuner dialog box displays.
- 8 Click a tuning button, and holding the mouse button down, slide the tuning bar up and down. Observe the simulation change on the graph as the variables are tuned.
- 9 Slide the tuners to the values shown in the following figure, and observe the resulting response on the graph of the tuned circuit.





- 10 Click the **X** at the top right of the Variable Tuner dialog box to close it and click the **Tune Tool** button on the toolbar to deactivate the tuner.

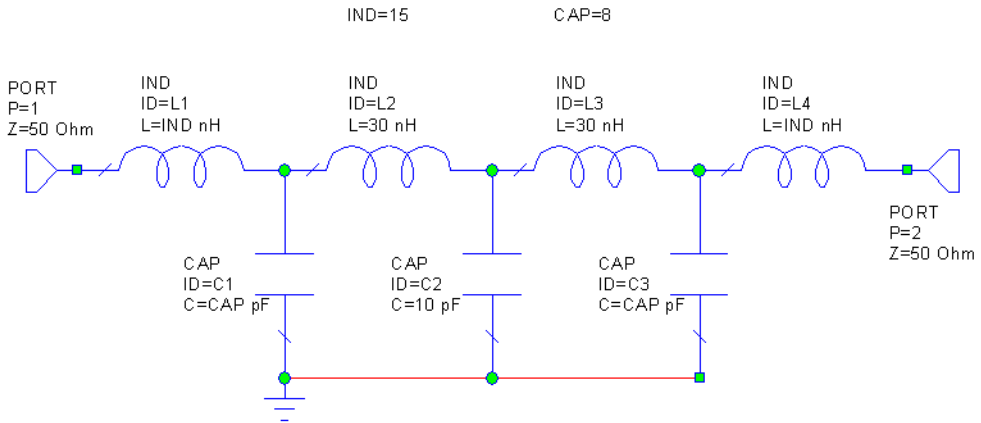
## Creating Variables

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:

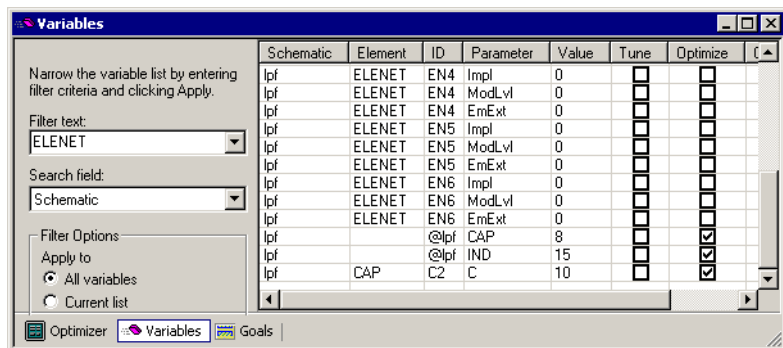
- 1 Click the schematic window to make it active.
- 2 Choose **Draw > Add Equation**.
- 3 Move the cursor into the schematic to display an edit box.
- 4 Position the edit box near the top of the schematic window and click to place it.
- 5 Type “**IND=15**” (without the quotes) in the edit box, and then 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 **Simulate > Optimize**. The Optimizer dialog box displays.
- 2 Click **Variables** at the bottom left of the dialog box to display the **Variables** tab.
- 3 Locate the IND and CAP variables in the **Parameter** column.
- 4 Click the box in the **Optimize** column in both the IND and CAP rows.
- 5 Click the box in the **Optimize** column in the CAP C2 row, then click the **X** at the top right of the dialog box to close it.

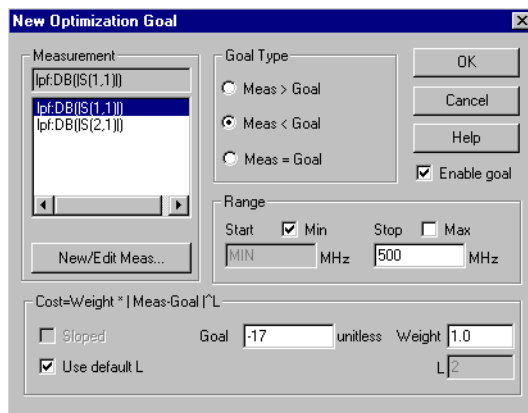


## Adding Optimization Goals

When you set an optimization goal, it is specified in the units set when the goal is created.

To add optimization goals:

- 1 In the Project Browser, right-click **Optimizer Goals** and choose **Add Opt 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” db as the return loss **Goal**, and then click **OK**.

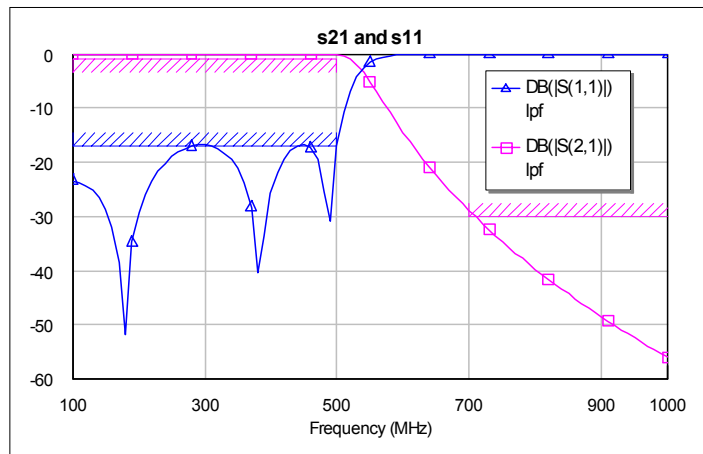
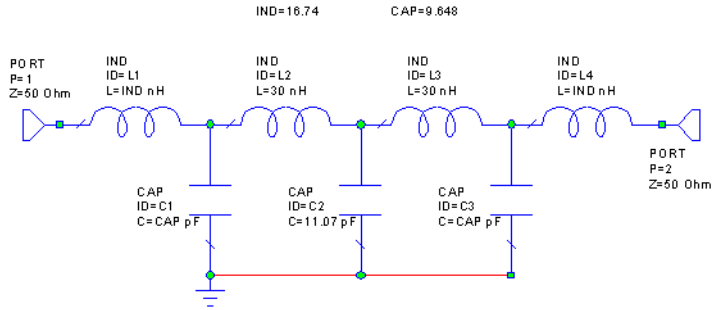


- 3 Repeat Step 1, then select **lpf:DB(|S(2,1)|)** as the **Measurement**, select **Meas > Goal** as the **Goal Type**, deselect **Max** under **Range** and type “500” as the **Stop** value, type “-1” db as the attenuation **Goal**, and then click **OK**.
- 4 Repeat Step 1, then select **lpf:DB(|S(2,1)|)**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

## Optimizing the Circuit

- 1 Choose **Simulate > Optimize**. The Optimizer dialog box displays.
- 2 Click the arrow to the right of **Optimization Methods** and select **Random (Local)**, 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.





# 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 building capabilities. The Layout view is actually another view of the schematic, and 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 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 User Guide*.

### Layout Tips and Tricks

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

Keystrokes	Layout Function
Press the <b>+</b> key	Zoom in
Press the <b>-</b> key	Zoom out
Press the <b>Home</b> key	Full view
Press the <b>Ctrl</b> 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 <b>Tab</b> key	Move shape with coordinate entry

Keystrokes	Layout Function
Press <b>Ctrl+Shift</b> while clicking on layered shapes	Cycle through layered shapes/elements and select them individually

## CREATING A LAYOUT FROM A SCHEMATIC

---

This example demonstrates how to use Microwave Office 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” in your *Program Files\AWR\AWR2004\Examples\Getting Started\Microwave Office\Layout* directory. You can use this example file as a reference.

To create a project:

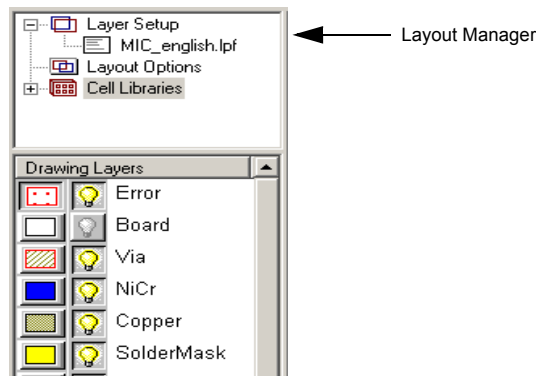
- 1 Choose **File > New Project**.
- 2 Choose **File > Save Project As**. The Save As dialog box displays.
- 3 Navigate to the directory in which you want to save the project, type “**layout\_example**” as the project name, and then click **Save**.

## 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 EMSight mappings.

To import an LPF:

- 1 Click the **Layout** tab at the lower left of the window 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\AWR2004** 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 Click the “MIC\_english.lpf” file and click **Open**. The following figure shows the Layout Manager.

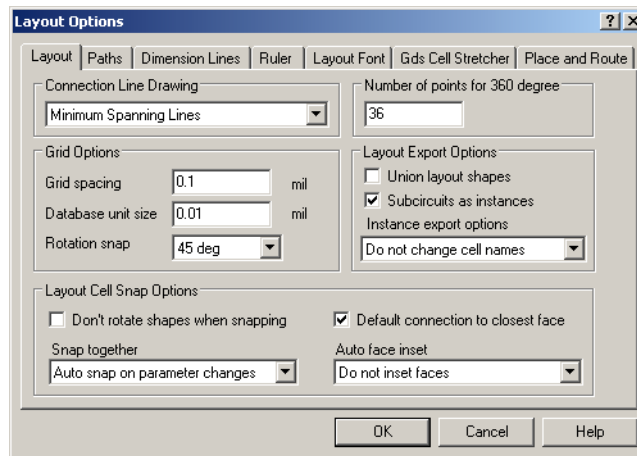


## Editing Database Units and Default Grid Size

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**, and then click **OK**.
- 3 In **Snap together**, select **Auto snap on parameter changes**.

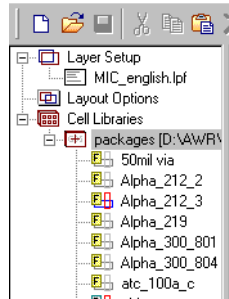


## Importing a GDSII Cell Library

Cell libraries are used in Microwave Office 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 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 **Read GDSII Library**.
- 2 Locate the program directory and double-click it to open it.
- 3 Double-click the **Examples** subdirectory, then double-click the **Getting Started, Microwave Office**, and **Layout** subdirectories.
- 4 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

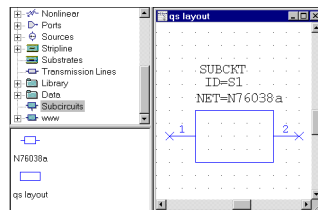
To import a data file:

- 1 In the Project Browser, right-click **Data Files** and choose **Import Data File**. The Add Data Document dialog box displays.
- 2 Locate the program directory and double-click it to open it.
- 3 Double-click the **Examples** subdirectory, then double-click the **Getting Started, Microwave Office, and Layout** subdirectories.
- 4 Select the “N76038a.s2p” file and then click **Open**.

## PLACING A DATA FILE IN A SCHEMATIC

To place a data file in a schematic:

- 1 Right-click **Circuit Schematics** in the Project Browser, choose **New Schematic**, and create a schematic named “**qs layout**”, then click **OK**.
- 2 In the Element Browser, expand the **Subcircuits** category.
- 3 Click the N76038a model and place it in the schematic window.

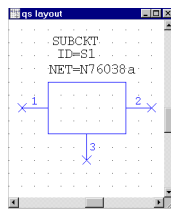


## CHANGING THE GROUND NODE OF A DATA FILE

You may occasionally need to access the ground node of a data file.

To expose the ground node of a transistor data file:

- 1 Double-click the subcircuit element in the schematic window. The Element Options dialog box displays.
- 2 Click the **Ground** tab.
- 3 Select **Explicit ground node**, then click **OK**.

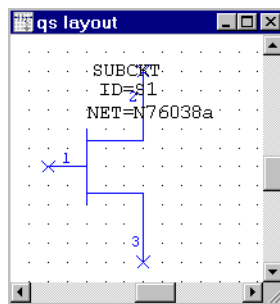


## Changing the Element Symbol

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**.



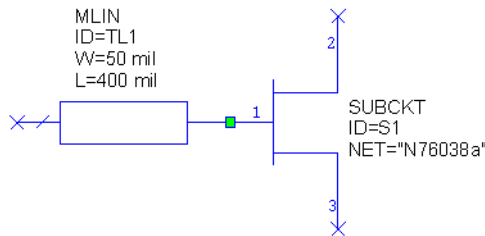
## Placing Microstrip Elements for Layout

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 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:

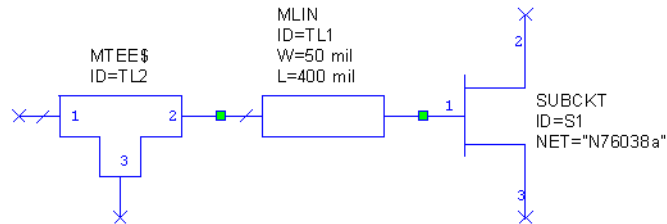
- 1 In the Element Browser, expand the **Microstrip** category, then click the **Lines** subgroup. Select the MLIN model and place it onto node 1 of the N7068a subcircuit in the schematic window.



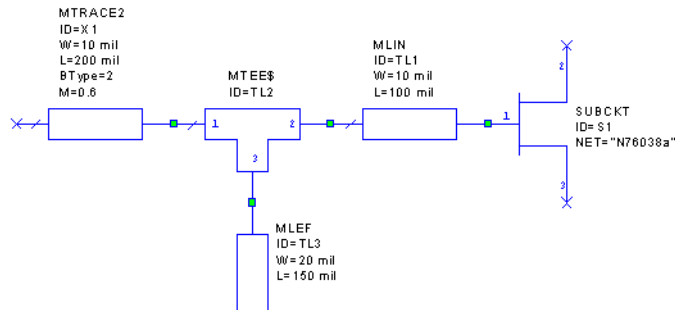
- 2 In the Microstrip category, click the **Junctions** subgroup. Select the MTEE\$ model and place it in the schematic window connected to the MLIN element as shown in the following figure.

**TIP:**

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 look-up table of EM-generated models. Thus, the name “MTEEX\$” is a microstrip tee junction based on an EM model look-up table that inherits its widths from the ports to which it connects.



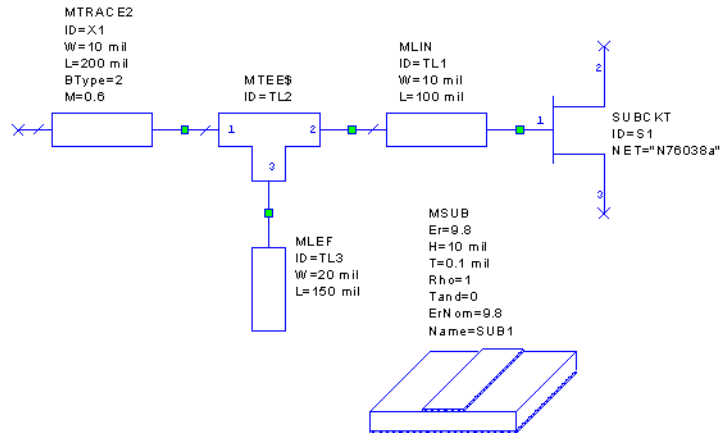
- 3 In the Microstrip category, click the **Lines** subgroup. Select the MTRACE2 model and place it in the schematic window onto node 1 of the MTEE\$ element.
- 4 Select the MLEF model in the same subgroup and place it in the schematic window. Right-click three times to rotate the element, then position it onto node 3 of the MTEE\$ element.
- 5 Double-click the MTRACE2 element in the schematic window to display the Element Options dialog box.
- 6 Edit the MTRACE2 parameters to match those shown in the following figure, then click **OK**.
- 7 Repeat step 6 for the MLIN and MLEF elements to match their parameters to those shown in the following figure.



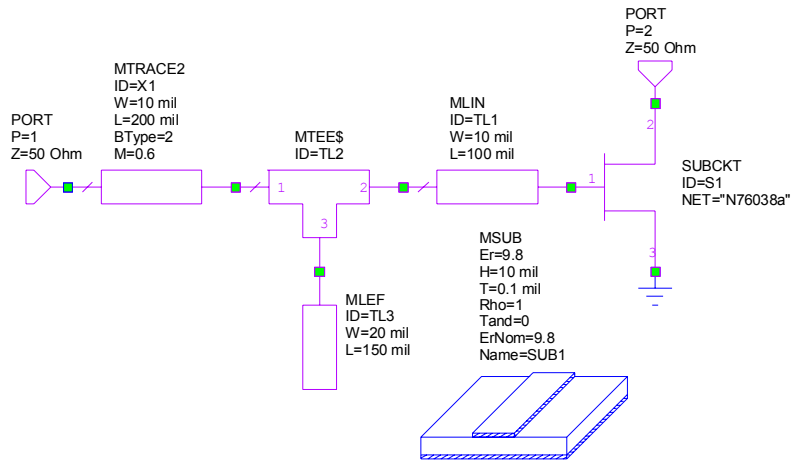
- 8 Click the **Substrates** category, then select the MSUB model and place it on the schematic window as shown in the following figure.



- 9 Double-click the MSUB element in the schematic window to display the Element Options dialog box. Edit the MSUB parameters to match those in the following figure, then click **OK**.



- 10 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.

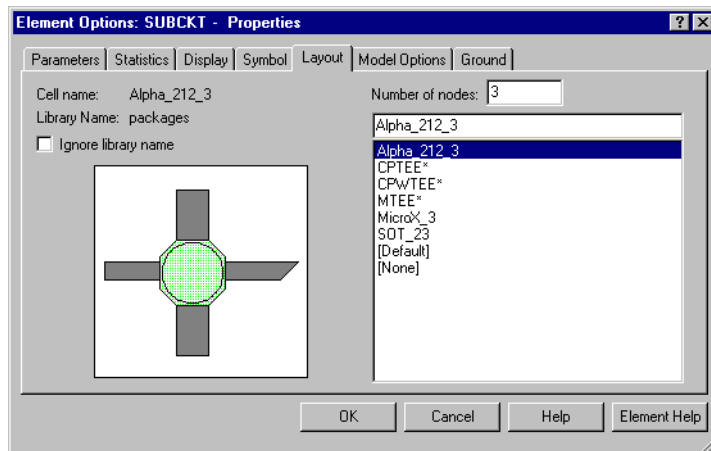


## Assigning an Artwork Cell to a Schematic Element

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 **Alpha\_212\_3** in the cell list at the right of the dialog box, then click **OK**.

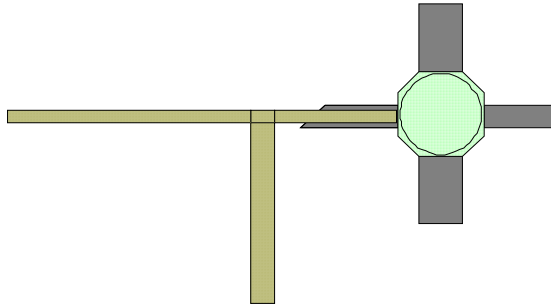


## Viewing a Layout

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 vice versa.

To view a layout:

- 1 Click the schematic window to make it active.
- 2 Choose **View > New Layout View** or click the **New Schematic Layout View** button on the toolbar to view a layout representation. The layout displays in a layout window.
- 3 Choose **Edit > Select All** to select all of the layout cells.
- 4 Choose **Edit > Snap Together** to snap all of the faces of the artwork cells together.

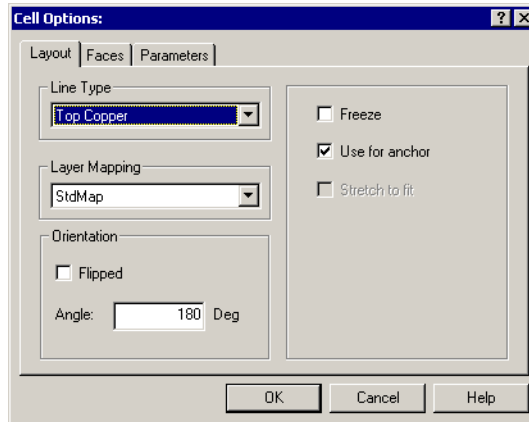


## Anchoring a Layout Cell

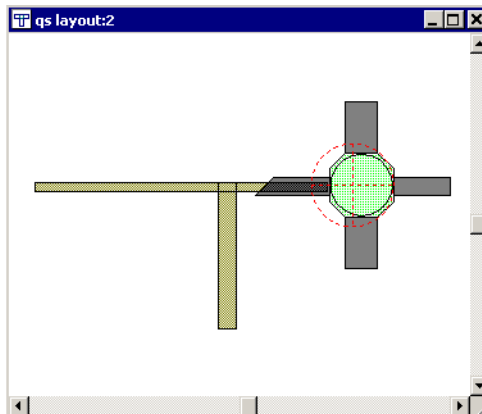
Layout cells have various properties that determine the connectivity of each cell in the layout view. One of the important properties is anchoring. Anchoring a layout cell holds the cell in place so that it cannot be moved by snapping functions. An anchored layout cell is typically used to define a reference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

To anchor a layout cell:

- 1 Select the “Alpha\_212\_3” artwork cell. Right-click, and choose **Shape Properties** to display the Cell Options dialog box.



- 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.



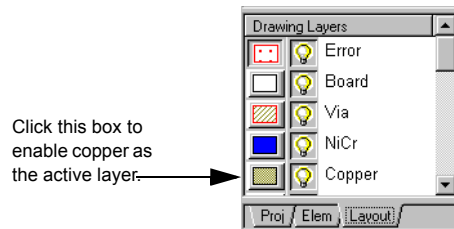
## Creating an Artwork Cell

To create an artwork cell:

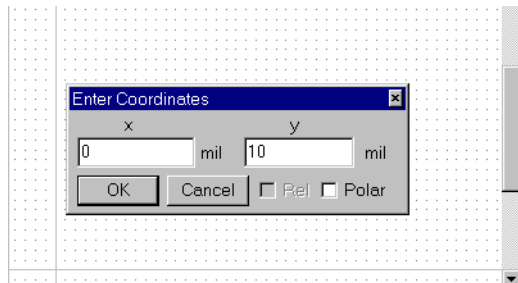
- 1 Click the **Layout** tab to activate the Layout Manager.
- 2 Right-click **packages** and choose **New Layout Cell**. The Create New Layout Cell dialog box displays.
- 3 Name the cell "**chip cap**" and click **OK**. A drawing window displays in the workspace.



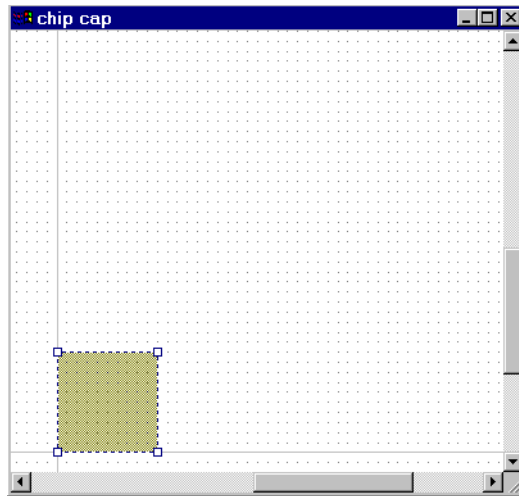
- 4 Click the **Set Grid Snap Multiple** button on the toolbar and set it to 10x. (Ensure that the Schematic Layout toolbar is displayed by right-clicking on the toolbar and selecting **Schematic Layout**.)
- 5 Click the chip cap package in the upper Layout Manager window, and then click the Copper box in the left column of the lower pane to enable copper as the active layer, as shown in the following figure. (Do not click the light-bulb, as it hides or displays the drawing layer.)



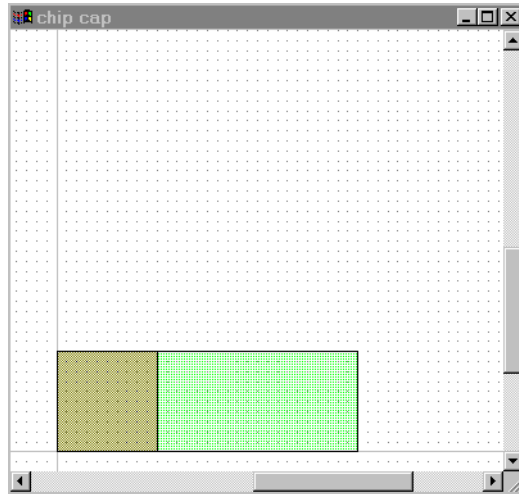
- 6 Choose **Draw > Rectangle**.
- 7 Move the cursor into the drawing window, then press the **Tab** key. The Enter Coordinates dialog box displays.
- 8 Type the values “0” and “10” in **x** and **y**, respectively, and click **OK**.



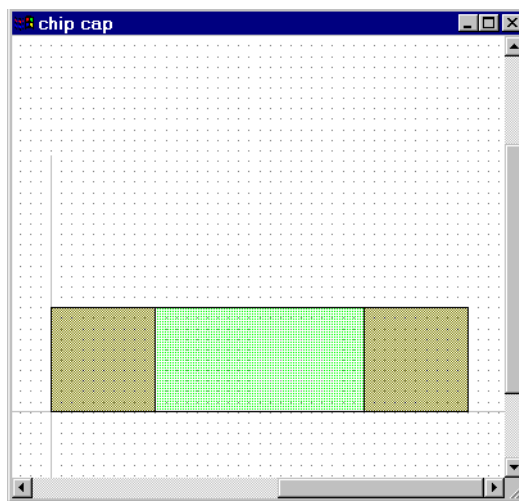
- 9 Press the **Tab** key again to display the Enter Coordinates dialog box.
- 10 Type the values “10” and “-10” in **dx** and **dy**, respectively, and click **OK**. The following figure shows the resulting drawing. (You may need to adjust your view by choosing **View > Zoom In** or **View > Zoom Out**.)



- 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. The Enter Coordinates dialog box displays.
- 15 Type the values “**10**” and “**10**” in **x** and **y**, respectively, and click **OK**.
- 16 Press the **Tab** key 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.

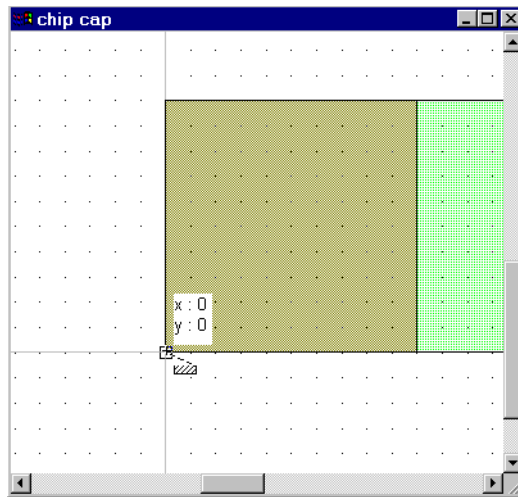


## ADD PORTS TO AN ARTWORK CELL

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.

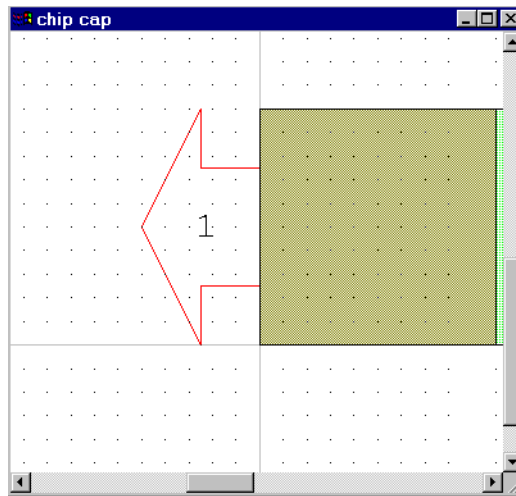
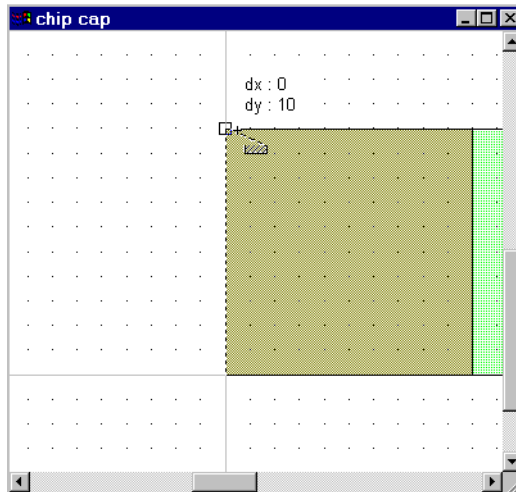
To add ports to an artwork cell:

- 1 Choose **Draw > Cell Port**.
- 2 Move the cursor into the “chip cap” window. Press and hold the **Ctrl** key while you move the cursor over the bottom left vertex of the square until a square symbol displays on the vertex. Do not release the **Ctrl** key.

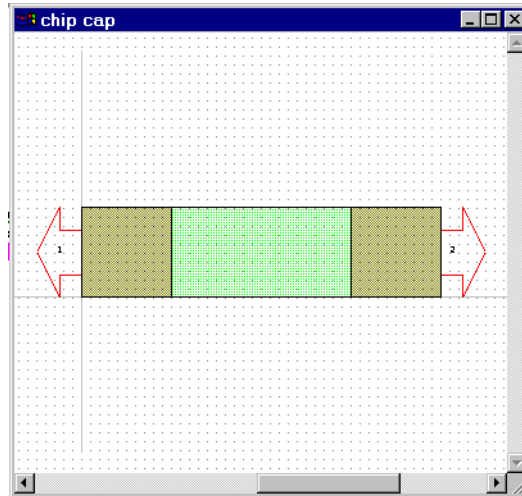


- 3 With the **Ctrl** key still pressed, click and hold down the mouse button while moving the cursor to the top vertex, until another square displays on that vertex. Release the mouse button and the **Ctrl** key.





- 4 Repeat steps 1 through 3 to place a port on the opposite side of the drawing, but start at the top vertex and draw down.

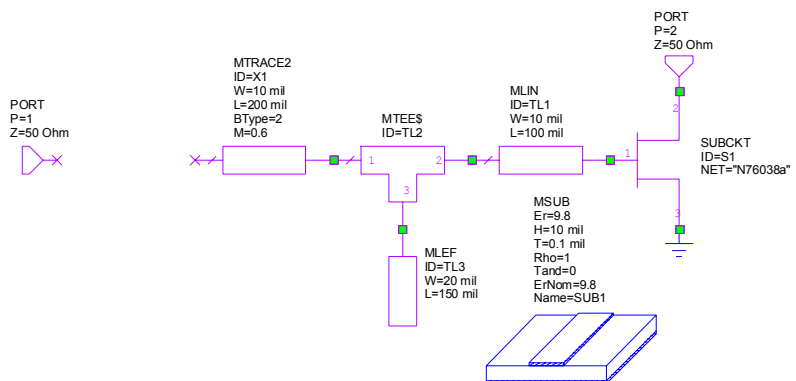


- 5 Click the **X** at the top right of the “chip cap” window. A dialog box asks if you want to save the cell edits. Click **Yes** to save.

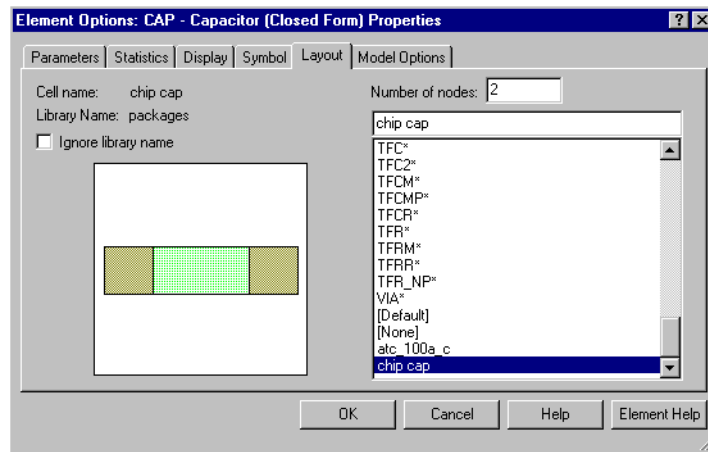
## Editing the Schematic and Assigning a Chip Cap Cell

To edit the schematic and assign a chip cap cell:

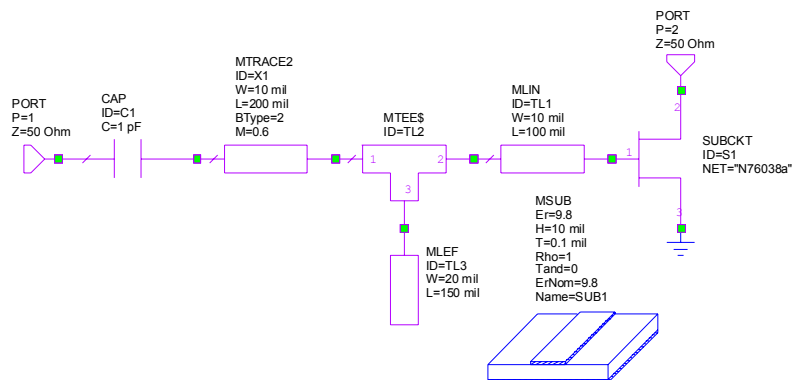
- 1 Click PORT 1 in the schematic window.
- 2 Press and hold the **Ctrl** key while you drag the port away from the MTRACE2 element, as shown in the following figure.

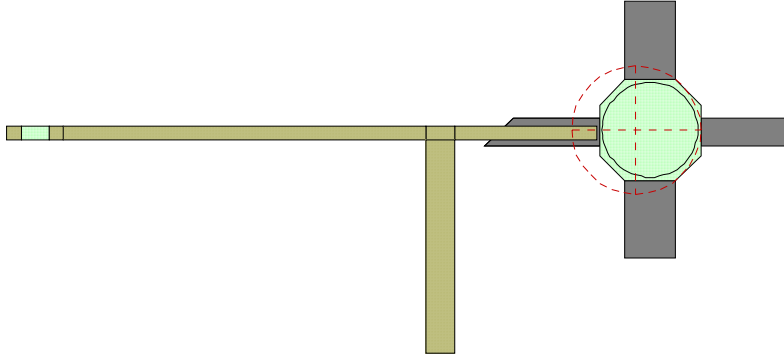


- 3 In the Element Browser, expand the **Lumped Element** category, then click the **Capacitor** subgroup. Select the CAP model, and place it in the schematic window between PORT 1 and the MTRACE2 element.
- 4 Double-click the CAP C1 element in the schematic window. The Element Options dialog box displays.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



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



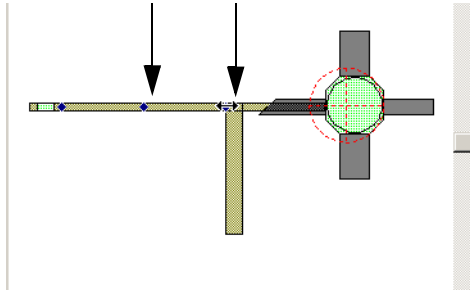


## Routing the MTRACE2 Element in Layout

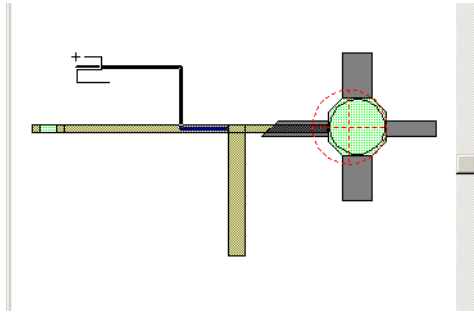
The MTRACE2 element is a special element that you can edit in the layout view to route a microstrip line.

To route the MTRACE2 element:

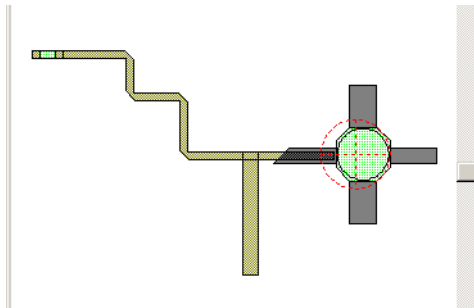
- 1 Double-click the MTRACE2 element in the Layout View to activate the blue grab diamonds.
- 2 Move the cursor over the right-most diamond until a double arrow symbol displays. Double-click to activate the routing tool.



- 3 Move the routing tool to another point and click to place. (Right-click to delete the last point; press the **Esc** key to cancel the activity.)



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

**TIP:**

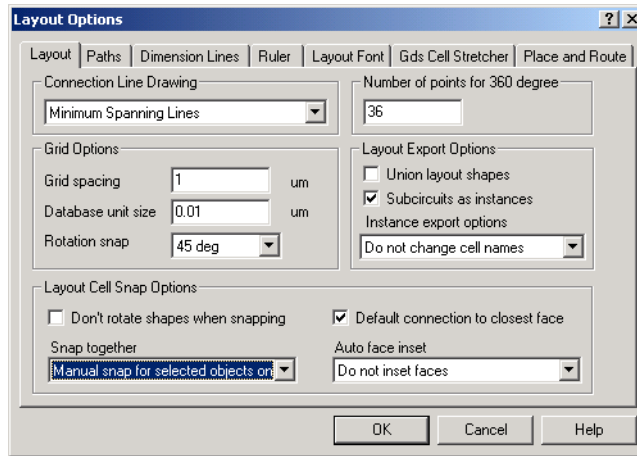
MLIN is a straight element with a width you can change in the layout. You can edit the MTRACE2 elements in the layout to create jogs and bends and chamfered corners. You can edit the MCTRACE element to create jogs and bends with rounded corners.

## 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.

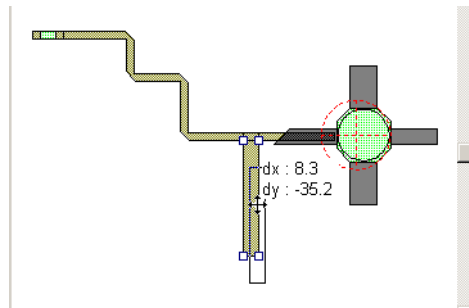
To specify the snapping options:

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

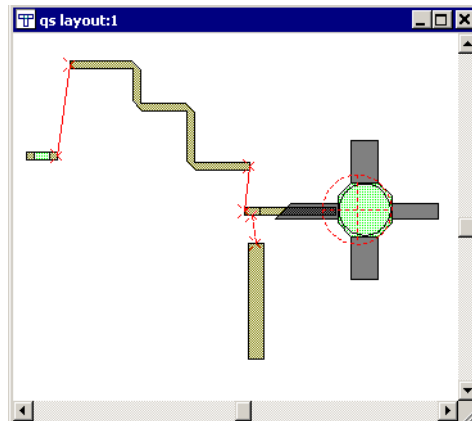


To separate the layout cells so the change in snapping options is viewable:

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



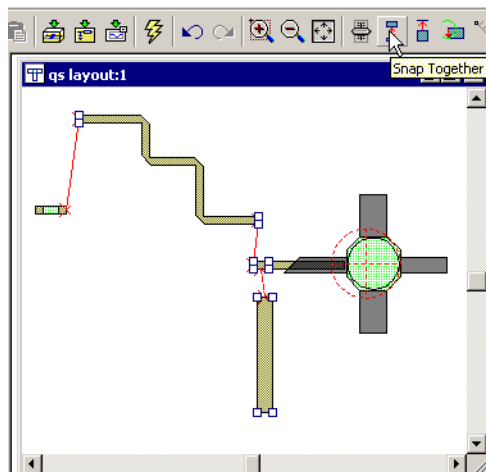
- 4 Repeat step 3 with the MTRACE2 element and the chip cap cell. Position the layout cells as shown in the following figure.



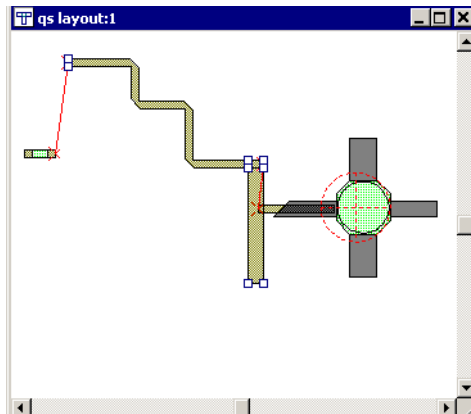
The red lines indicate that the faces of the layout cells are not snapped together.

To snap a selected set of layout cells together:

- 5 Hold down the **Shift** key and select the MLEF, MTRACE2, and MTEE\$ layout cells in the layout window.

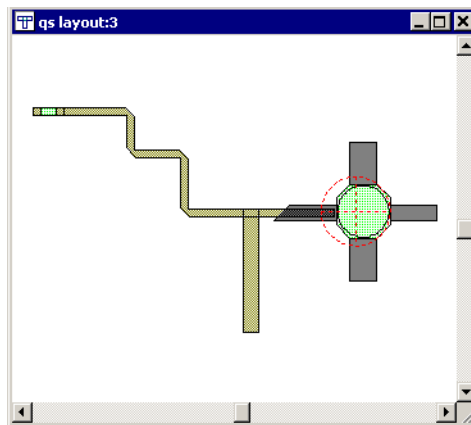


- 6 Click the **Snap Together** button on the toolbar. Observe that the chip cap layout cell and MLIN layout cell are not snapped together.



To snap all of the faces together:

- 7 Type **Ctrl+A** to select all of the layout cells.
- 8 Click the **Snap Together** button on the toolbar. The layout displays as shown in the following figure.

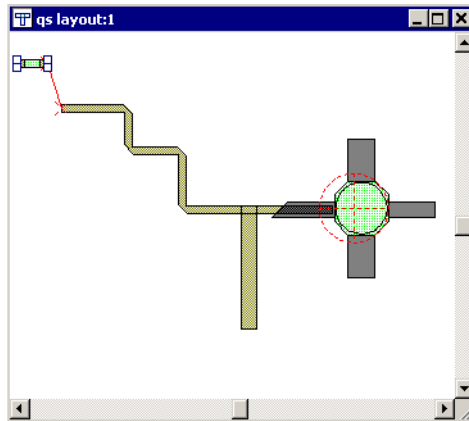


The “snap to fit” function finishes the routing of an MTRACE2 layout cell to a specified adjacent layout cell. In this example, the chip cap layout cell is moved and MTRACE2 re-routes to snap to the chip cap face.

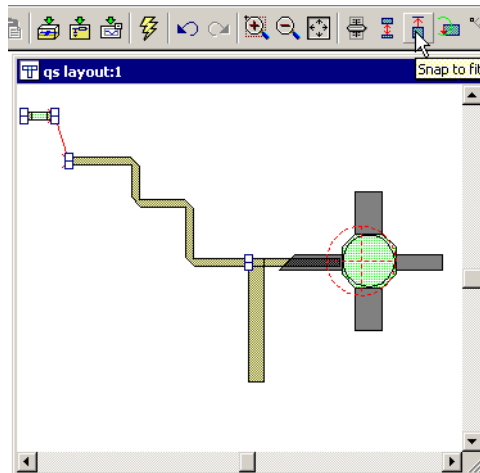
To “snap to fit” MTRACE2 to the chip cap:

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

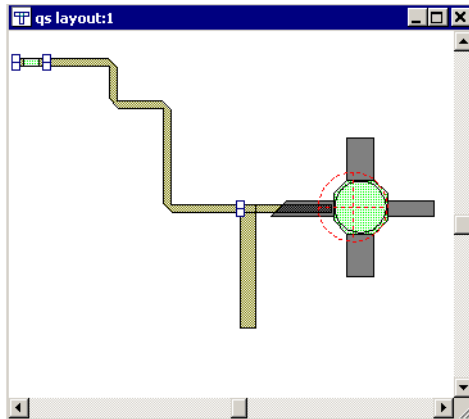




- 10 Select the MTRACE2 layout cell, hold down the **Shift** key and select the chip cap artwork cell.



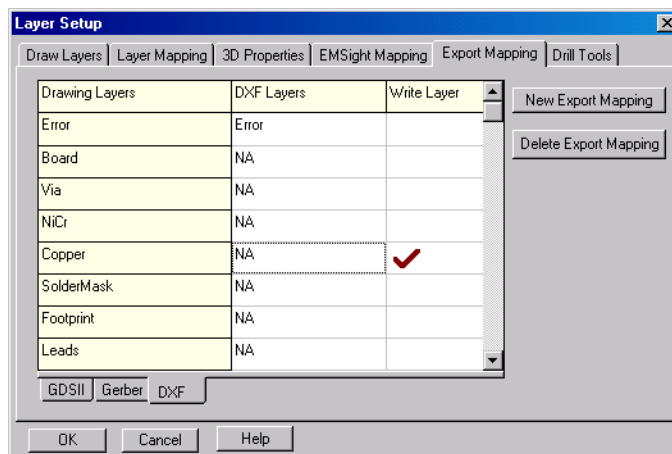
- 11 Click the **Snap to Fit** button on the toolbar. The MTRACE2 routes to snap to the chip cap artwork cell as shown in the following figure.



## Exporting the Layout

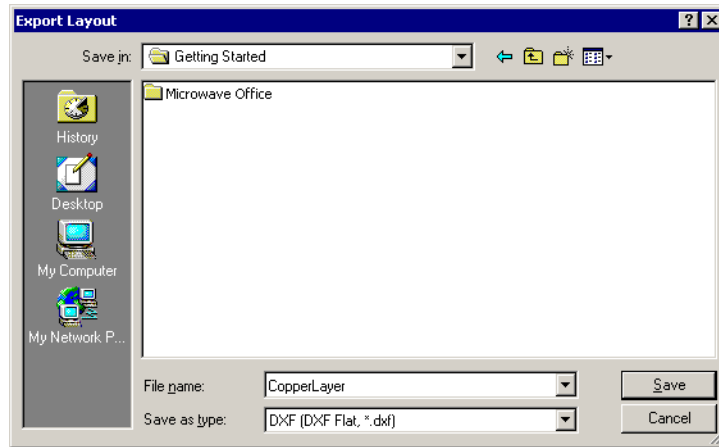
To export a layout:

- 1 Choose **Options > Drawing Layers** to specify the file layers to export. The Layer Setup dialog box displays.
- 2 Click the **Export Mapping** tab.
- 3 Click the **DXF** tab at the bottom of the dialog box, and in the **Write Layer** column, deselect all of the drawing layers except the copper layer, then click **OK**.



- 4 Choose **Layout > Export**. 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 your work by choosing **File > Save Project**.



# 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. For more detailed information, see the *Microwave Office User Guide*.

## HARMONIC BALANCE IN MICROWAVE OFFICE

---

Microwave Office 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 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 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 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 a 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

### Creating a New Project

The example you create in this chapter is available in its complete form as “nonlinear\_example.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\Microwave Office\Nonlinear* directory. You can use this example file as a reference.

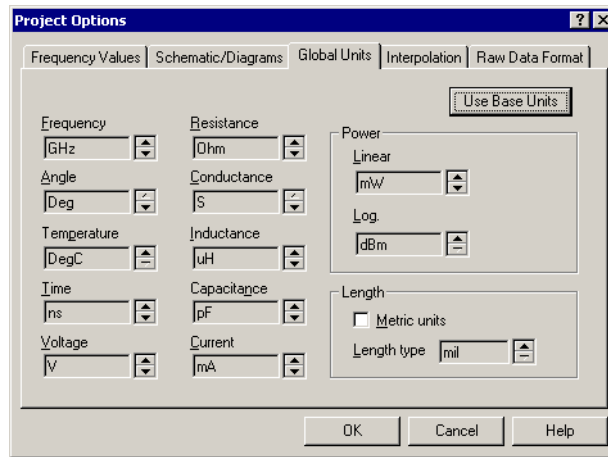
To create a project:

- 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.

## Creating a Schematic

To create a schematic:

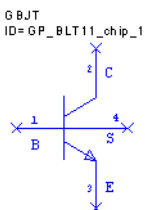
- 1 Choose **Project > Add Schematic > New Schematic**. The Create New Schematic dialog box displays.
- 2 Type **"IV Curve"**, and click **OK**. A schematic window displays in the workspace.

## PLACING A NONLINEAR MODEL FROM THE LIBRARY

To place a nonlinear model:

- 1 Click the **Elem** tab at the lower left of the window to display the Element Browser.

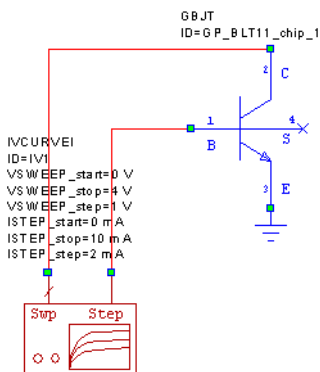
- Expand the **Library** category, then expand the **Nonlinear** and **Getting Started** subgroups. Click the **GBJT** subgroup, then select the BLT11\_chip model and place it as shown in the following figure.



### PLACING AN IV CURVE METER ON THE NONLINEAR ELEMENT

To place an IV curve meter element with stepped current:

- Under **Circuit Elements**, expand the **MeasDevice** category, then click the **IV** subgroup. Select the IVCURVEI model and place it as shown on the following schematic.
- 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.
- Repeat step 2 to connect the Swp node of IVCURVEI to node 2 of the GBJT transistor.
- 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**.

Name	Value	Unit	Tune	Opt	Limit	Lower	Upper	Description
ID	IV1							Element ID
VSWEEP_start	0	V				0	0	Swept voltage start value
VSWEEP_stop	15	V				0	0	Swept voltage stop value
VSWEEP_step	0.1	V				0	0	Swept voltage step value
ISTEP_start	0	mA				0	0	Stepped current start value
ISTEP_stop	10	mA				0	0	Stepped current stop value
ISTEP_step	1	mA				0	0	Stepped current step value

Element ID

☒ Enable element    Part Number

Hide Secondary

OK Cancel Help Element Help

## ADDING AN IV CURVE MEASUREMENT

To create a graph and add an IV curve measurement:

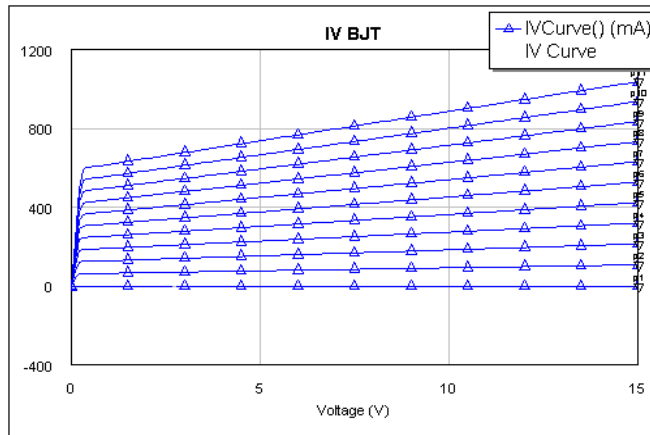
- 1 In the Project Browser, right-click **Graphs** and choose **Add Graph**. The Create Graph dialog box displays.
- 2 Type “**IV BJT**” in **Graph Name**, select **Rectangular** as the **Graph Type**, and click **OK**. The graph displays in the workspace.
- 3 Right-click the “**IV BJT**” graph in the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



### TIP:

You can disable parameter markers by right-clicking on the graph and choosing **Properties**. On the **Markers** tab of the Graph Properties dialog

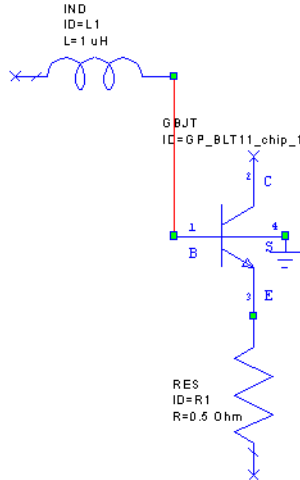
box, clear the **Param markers enabled** and **Param markers in legend** check boxes.



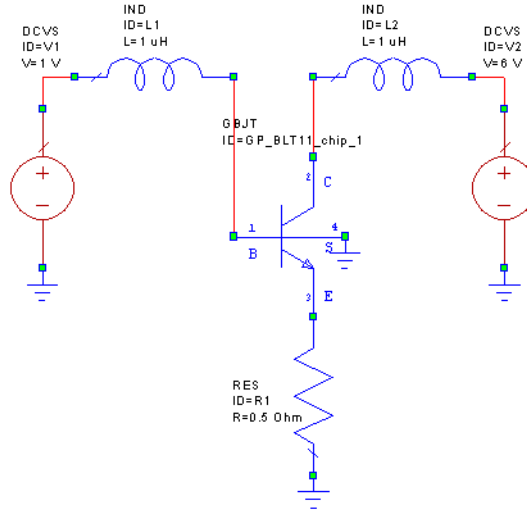
## Creating a Bias Circuit

To create a DC bias circuit:

- 1 Choose **Project > Add Schematic > New Schematic**. The Create New Schematic dialog box displays. Type “**DC Bias**” and click **OK**.
- 2 In the Element Browser, expand the **Library** category, then expand the **Nonlinear** and **GettingStarted** subgroups. Click the **GBJT** subgroup, then select the BLT11\_chip model and place it on the schematic as shown in the following figure.
- 3 Expand the **Lumped Element** category, then click the **Inductor** subgroup. Select the IND model, and place it above and to the left of the GBJT transistor as shown in the following figure.
- 4 Place the cursor on node 1 of the GBJT transistor. The cursor displays as a wire coil symbol. Click, then drag the cursor to the right node of IND, and click to place the wire.
- 5 Double-click the IND model and set the L parameter to “1”, then click **OK**.
- 6 Click the **Resistor** subgroup under **Lumped Element**. Select the RES model and place it as shown in the following figure after right-clicking once to rotate the element.



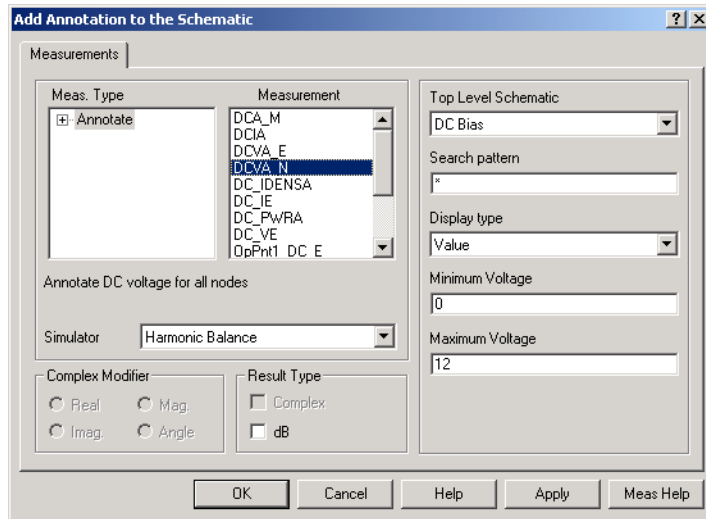
- 7 Double-click the RES model and set the R parameter to “0.5”, then click **OK**.
- 8 Click the **Ground** button on the toolbar and position the ground on the bottom of RES R1 as shown in the following figure.
- 9 Repeat step 8, positioning the ground on node 4 of the GBJT transistor as shown in the following figure.
- 10 Expand the **Sources** category, then click the **DC** subgroup. Select the DCVS model and place it as shown in the following figure.
- 11 Click the **Ground** button on the toolbar and position the ground on the open end of DCVS V1 as shown in the following figure.
- 12 Double-click the DCVS model and set the V parameter to “1”, then click **OK**.
- 13 Click IND L1 in the schematic window. Press **Ctrl+C**, then **Ctrl+V** to copy and paste it. Connect the new IND element to node 2 of the GBJT model as shown in the following figure.
- 14 Copy the DCVS model and place the copy on the open node of IND L2 as shown in the following figure.
- 15 Double-click the DCVS V2 model and set the V parameter to “6”, then click **OK**.
- 16 Click the **Ground** button on the toolbar and position the ground on the negative node of DCVS V2 as shown in the following figure.



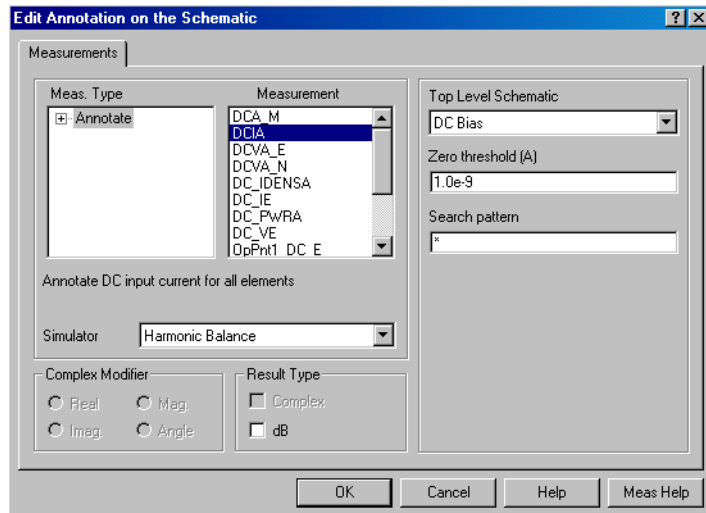
### ADDING SCHEMATIC BACK ANNOTATION

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

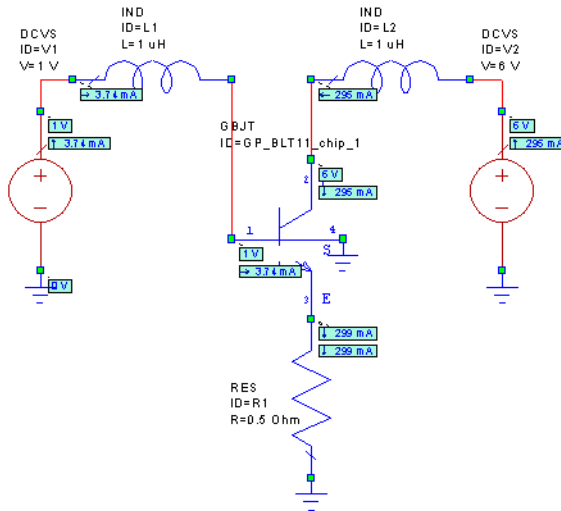
- 1 Right-click the “DC Bias” schematic in the Project Browser and choose **Add Annotation**. The Add Annotation to the Schematic dialog box displays.
- 2 Specify a voltage measurement using the settings in the following figure, then click **OK**.



- 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:

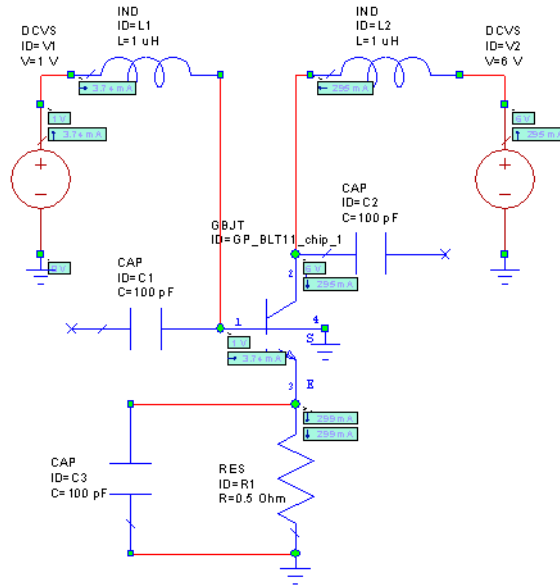
- 1 Click the “DC Bias” schematic window in the workspace to make it active.
- 2 In the Element Browser, expand the **Lumped Element** category, then click the **Capacitor** subgroup. Select the **CAP** model and connect it to node 1 of the GBJT transistor as shown in the following figure.
- 3 Double-click 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:

- 5 Copy the CAP C1 model and place the copy (CAP C3) to the left of RES R1 after right-clicking once as shown in the following figure.
- 6 Connect the top node of CAP C3 to node 3 of the GBJT transistor.

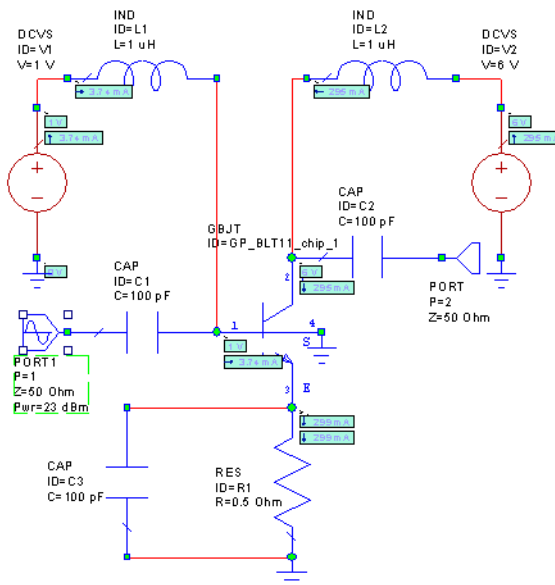
- 7 Connect a wire between the bottom node of CAP C3 and ground.



To add a harmonic balance port:

- 1 In the Element Browser, expand the **Ports** category, then click the **Harmonic Balance** subgroup. 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 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:

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

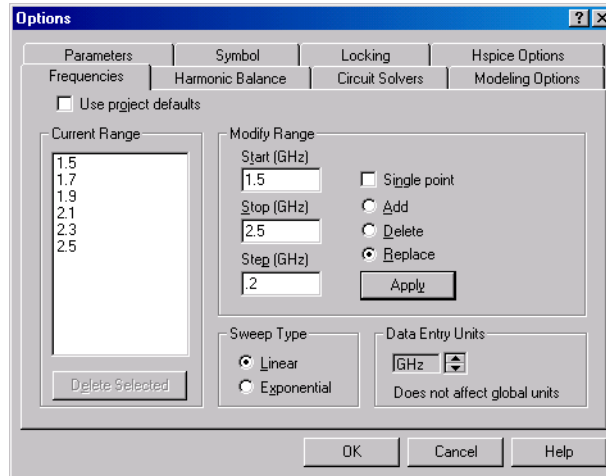
You select the frequency sweep used by the simulator in the Add/Modify Measurement dialog, as shown in “Adding a Large Signal Reflection Coefficient Measurement” on page 6-13. The following step defines the frequency sweep using a document sweep, which is the simplest and most convenient means.

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 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.

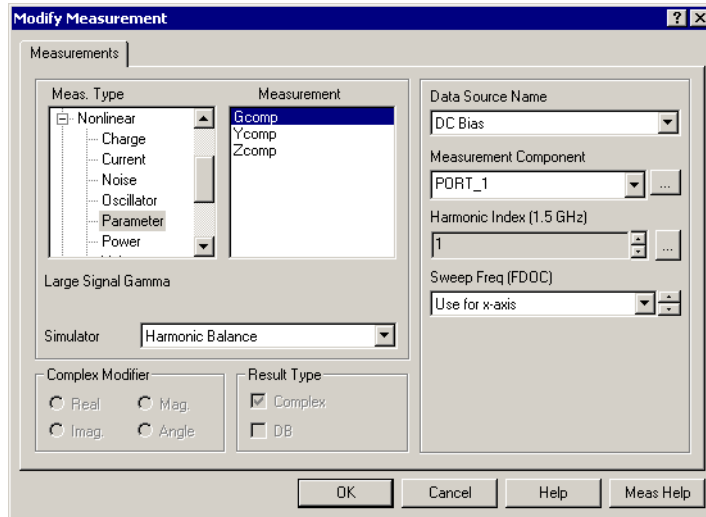
To create a Smith Chart:

- 1 In the Project Browser, right-click **Graphs** and choose **Add Graph**. The Create Graph dialog box displays.
- 2 Type **"Input reflection"** in **Graph Name**, select **Smith Chart** as the **Graph Type**, and click **OK**.

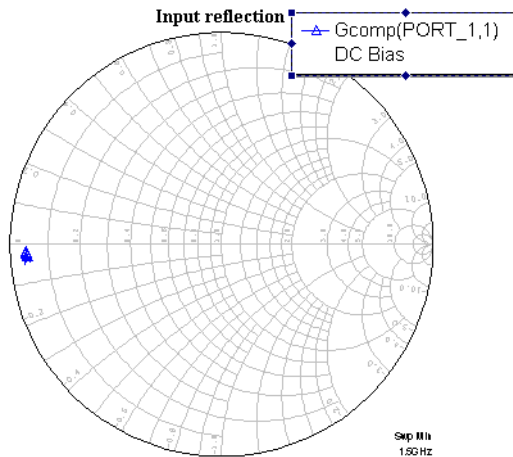
To add a large signal reflection measurement:

- 3 Right-click "Input reflection" under **Graphs** and choose **Add Measurement**. The Add Measurement dialog box displays.
- 4 Select **Nonlinear > Parameter** as the **Meas. Type**, **Gcomp** as the **Measurement**, and **DC Bias** as the **Data Source Name**. Select **PORT\_1** as the **Measurement Component**, click the arrows to the right to select **1** in **Harmonic Index**, and then click **Apply** and **OK**.

Note the frequency sweep control which 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” on page 6-12.



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

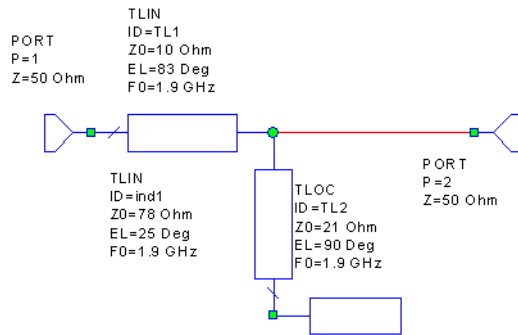


## Importing Schematics

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

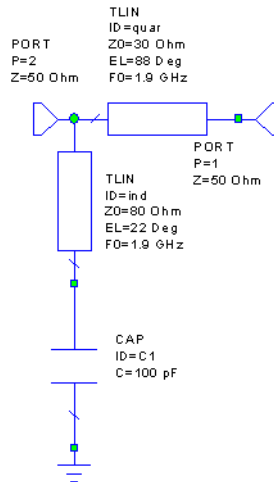
To import the input match schematic:

- 1 In the Project Browser, right-click **Circuit Schematics** and choose **Import Schematic**. The Open dialog box displays.
- 2 Locate the **C:\Program Files\AWR\AWR2004\Examples\Getting Started\Microwave Office\Nonlinear** directory and double-click it to open it.
- 3 Select the “input match.sch” file and click **Open** to import and open the schematic.



To import the output match schematic:

- 4 Right-click **Circuit Schematics** and choose **Import Schematic**.
- 5 Select the “output match.sch” file from the previously opened directory and click **Open** to import and open the schematic.

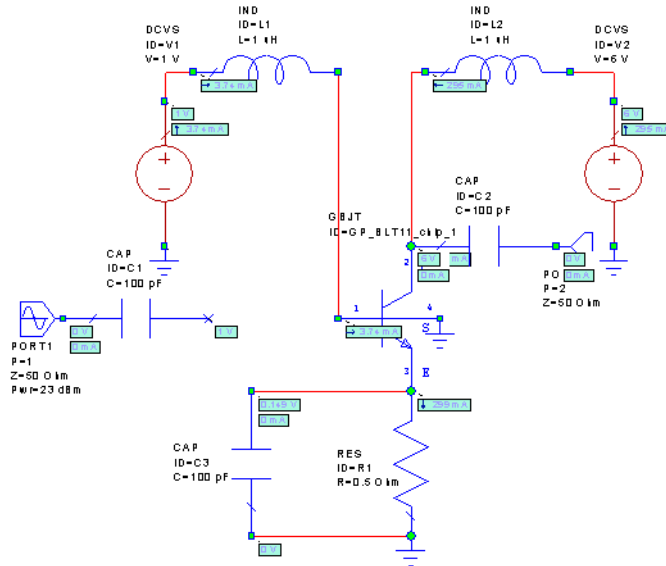


## Adding Subcircuits to a Schematic

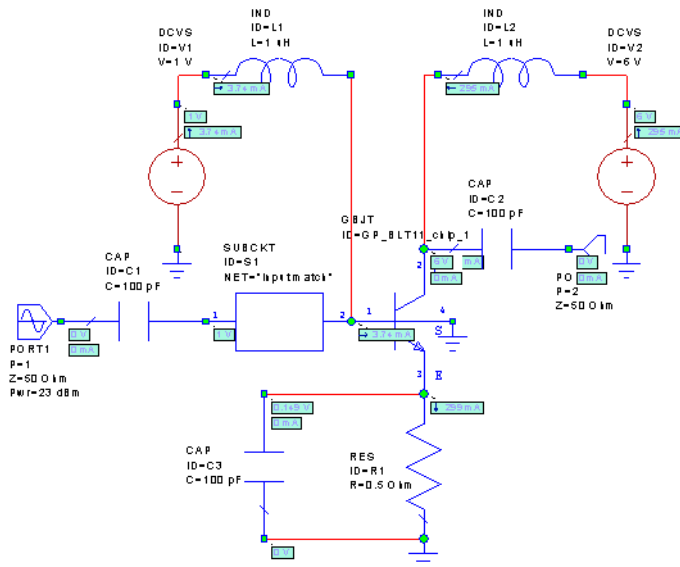
Whenever an n-port schematic is created or imported it automatically becomes an n-port subcircuit. You can use these subcircuits within other schematics to create circuit hierarchy.

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

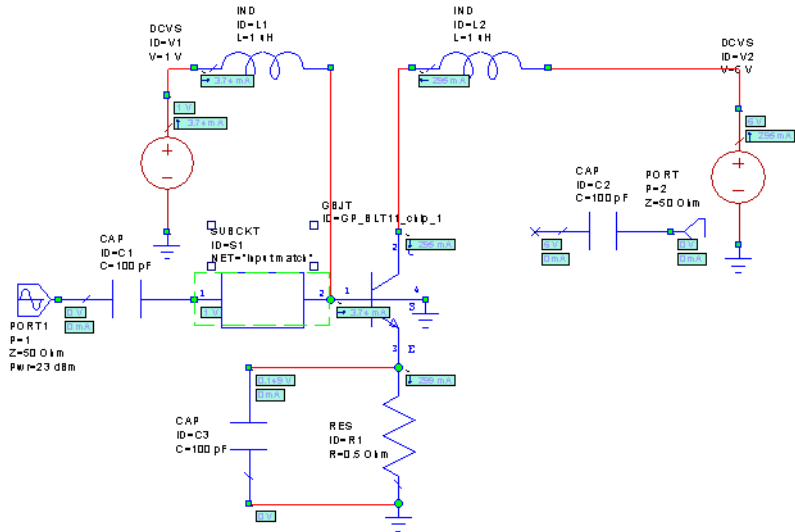
- 1 In the Project Browser under **Circuit Schematics**, double-click “DC Bias” to display the DC Bias schematic.
- 2 Click PORT1 in the schematic window, then press and hold the **Shift** key while clicking on CAP C1. Both PORT1 and CAP C1 should be selected (and are now considered one unit).
- 3 Press the **Ctrl** key and click once 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.



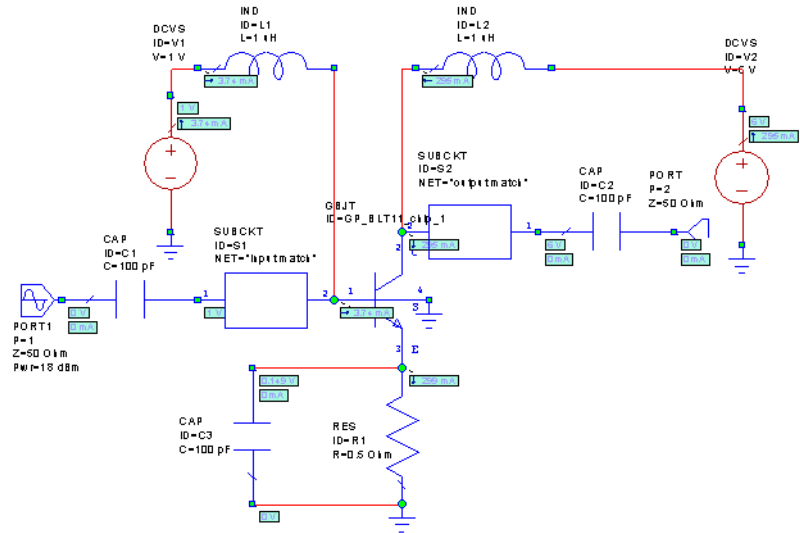
- 4 In the Element Browser, click the **Subcircuits** category then select the “input match” subcircuit and place it on the schematic between CAP C1 and node 1 of the GBJT transistor as shown in the following figure.



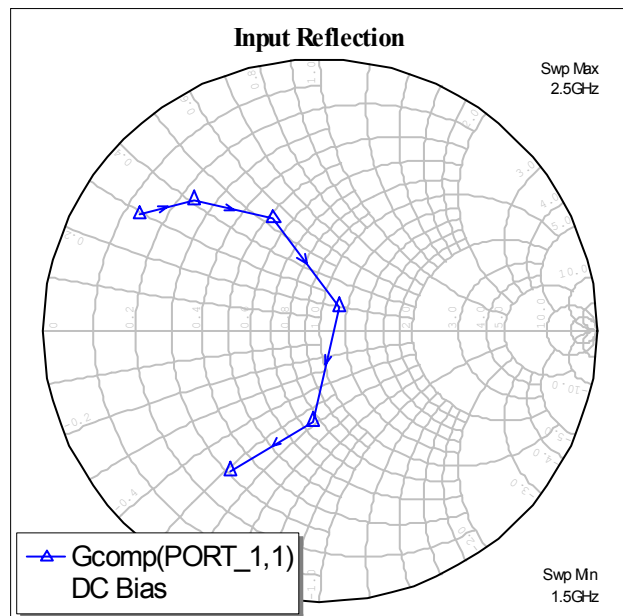
- 5 If the subcircuit nodes do not properly connect with the capacitor and transistor you may need to slightly move the elements until the proper connections are made.
- 6 Repeat steps 2 and 3 with PORT2 and CAP C2 as shown in the following figure.



- 7 Click the **Subcircuits** category then 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.



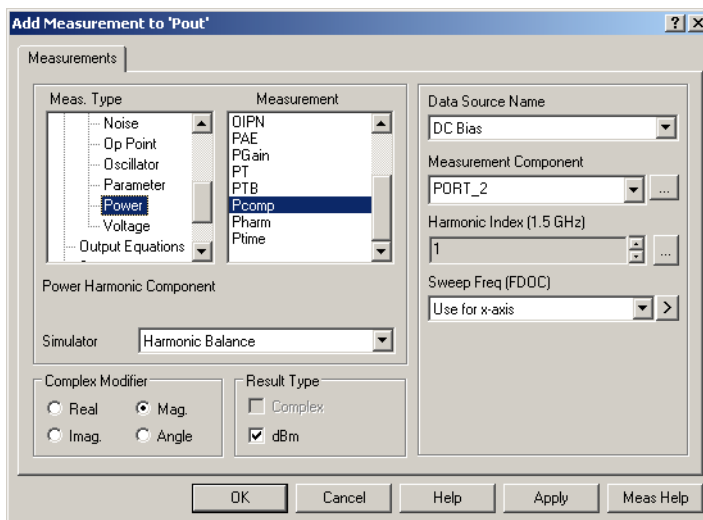
- 10 Choose **Simulate > Analyze**. The simulation response in the following Smith chart displays with the circuit matched.



## Creating a Pout vs. Frequency Measurement

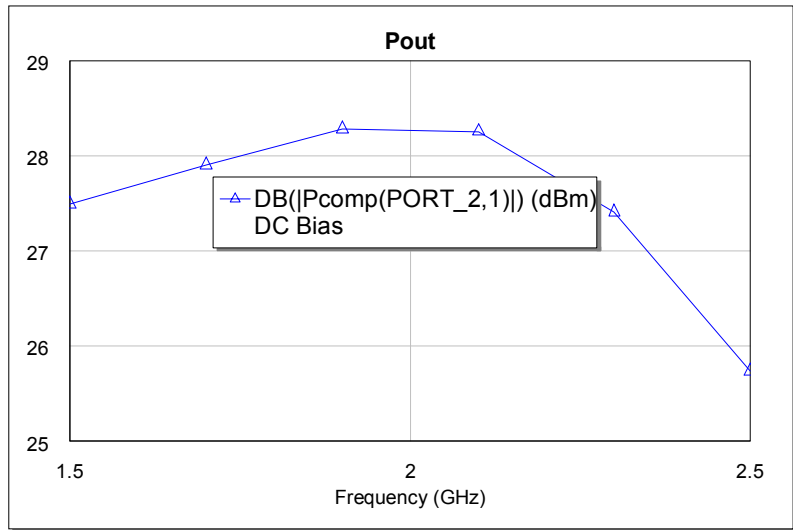
To create a graph and add a measurement (Pout versus frequency):

- 1 In the Project Browser, right-click **Graphs** and choose **Add Graph**. The Create Graph dialog box displays.
- 2 Type “**Pout**” in **Graph Name**, select **Rectangular** as the **Graph Type**, and click **OK**.
- 3 Right-click “Pout” 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.



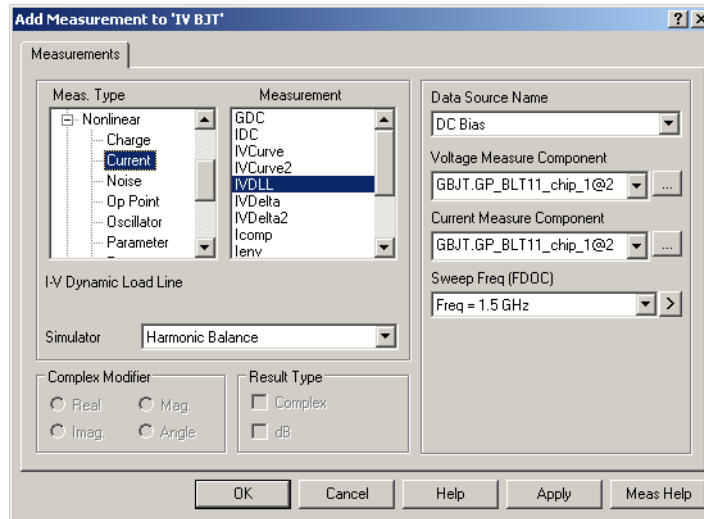


## Creating a Dynamic Load Line Measurement

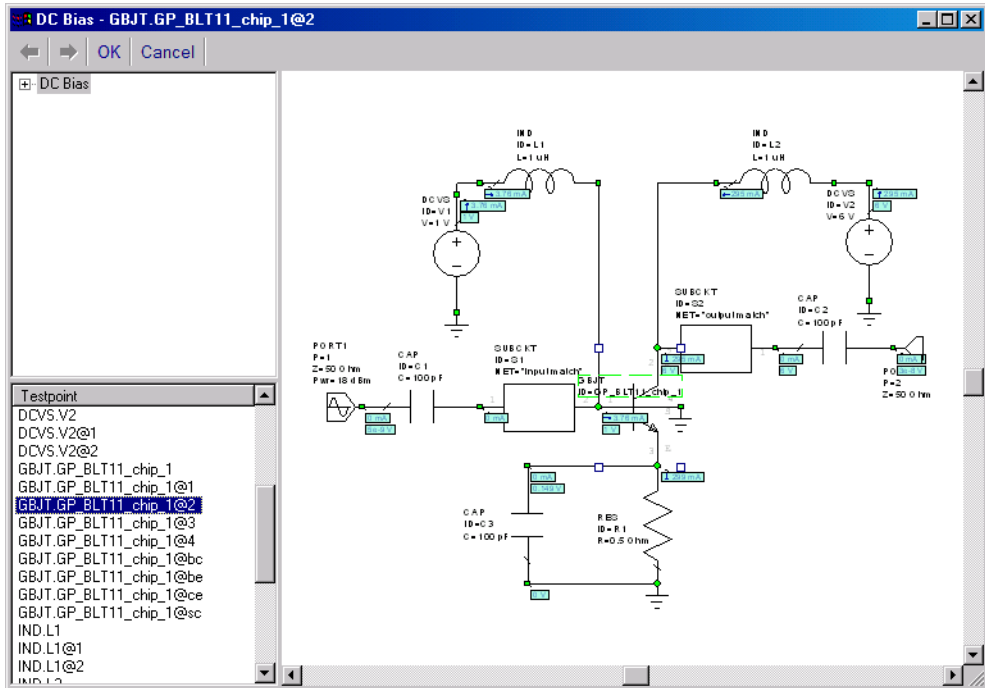
A dynamic load line measurement plots the large signal performance of the circuit superimposed on the IV curve of the device.

To create a dynamic load line measurement:

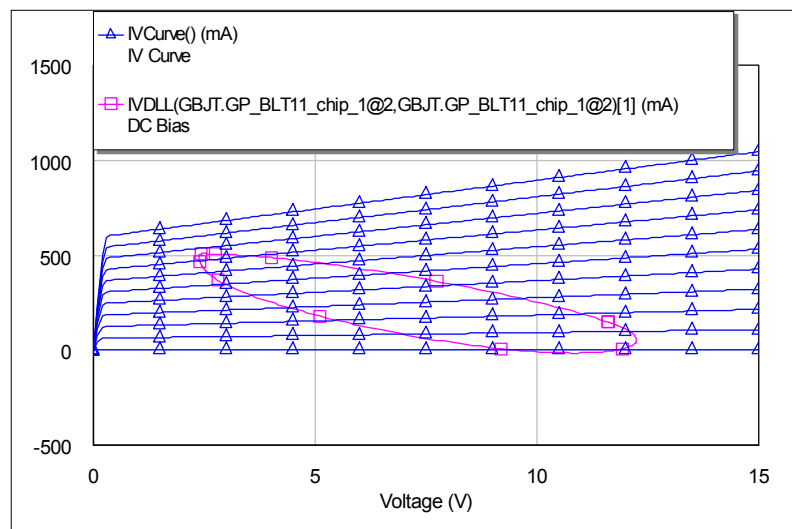
- 1 In the Project Browser, right-click “IV BJT” under **Graphs** and choose **Add Measurement**. The Add Measurement dialog box displays.
- 2 Create a measurement using the settings in the following figure, then click **OK**.



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.



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



## Copying a Schematic in the Project Browser

The next 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.

To duplicate the DC bias schematic:

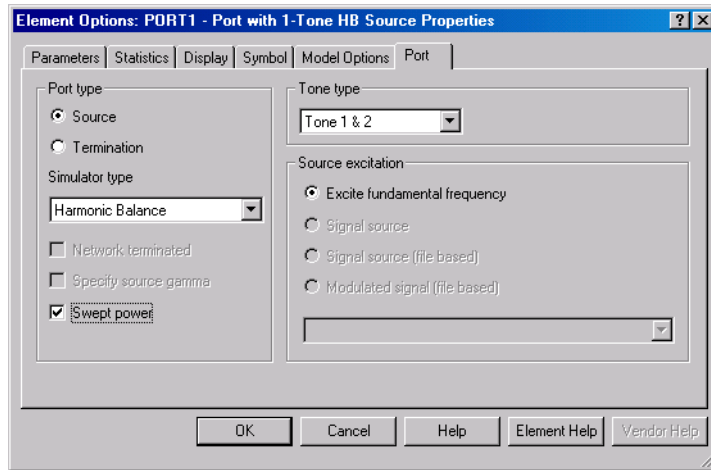
- 1 Click “DC bias” under **Circuit Schematics**, drag it up to the **Circuit Schematics** node and release the mouse button. A duplicate schematic named “Copy of DC” bias is created.
- 2 Right-click the “Copy of DC Bias” schematic and choose **Rename Schematic**. Rename the schematic to “**Two Tone Amp**” in the Rename Data Source dialog box, and then click **OK**.

### ADDING A TWO-TONE HARMONIC BALANCE PORT

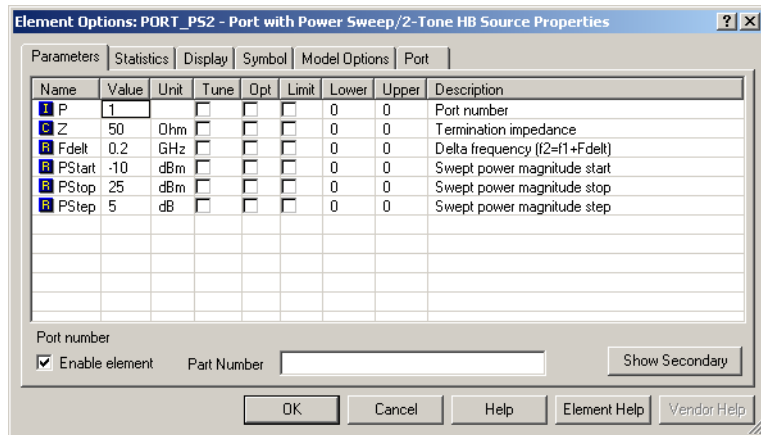
A common measurement used to characterize power amplifiers is a third-order intermodulation product versus swept power. To make this measurement, two closely-spaced tones must be injected into the input port.

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.



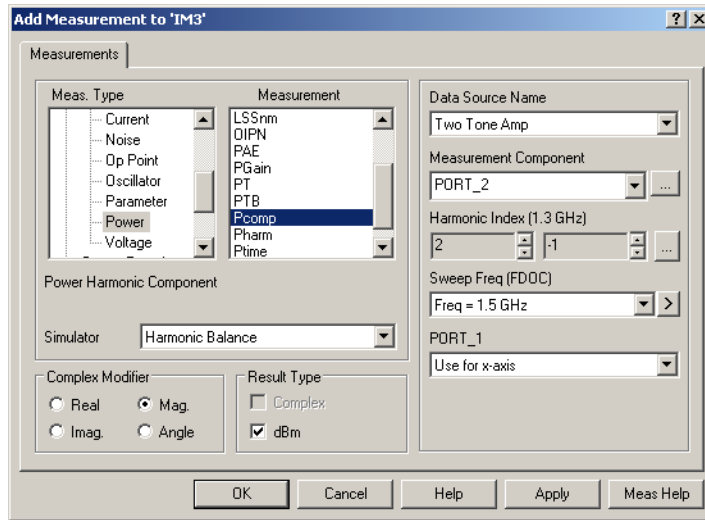
- 5 Click the **Parameters** tab and edit the parameters to the values shown in the following figure, then click **OK**.



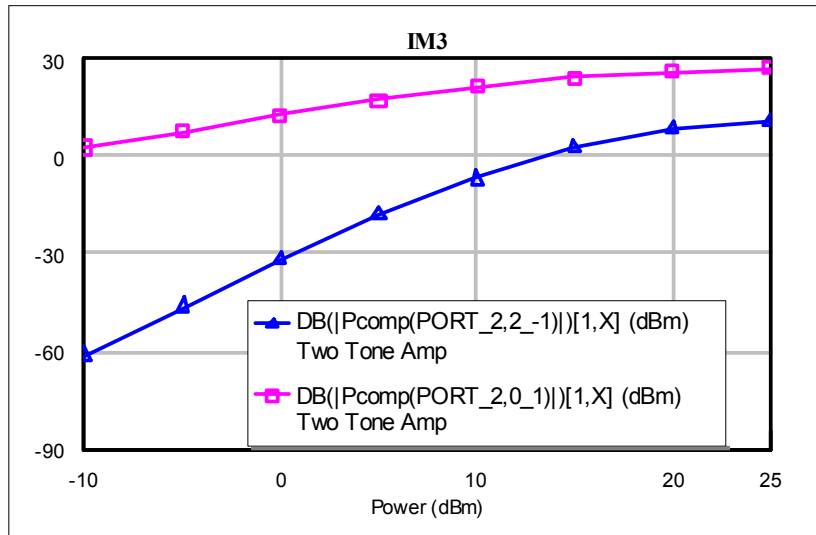
### ADDING A THIRD-ORDER INTERMODULATION MEASUREMENT

To add a third-order intermodulation product measurement:

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



- 3 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 multidimensional (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 “Copy of Two Tone Amp” schematic and rename it “**IP3**”.

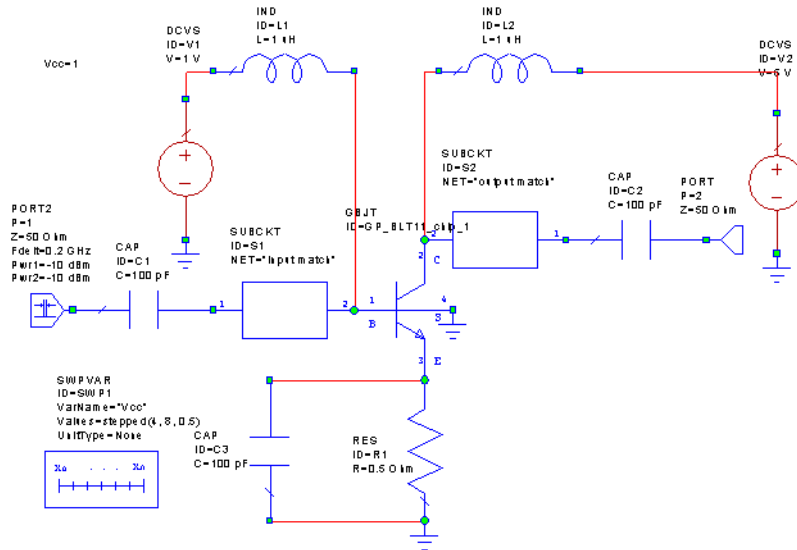
To change the port excitation to a fixed power:

- 3 Double-click Port\_PS2 P=1. 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.
- 4 Double-click the Pwr1 and Pwr2 parameter values of PORT2 P=1 and change both to “**-10**”.

To define a swept variable:

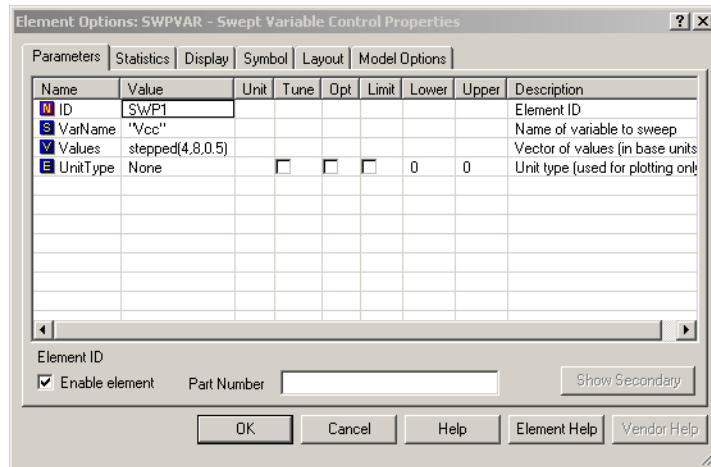


- 1 Choose **Draw > Add Equation**. You can also click the **Equation** button on the toolbar.
- 2 Move the cursor into the schematic to display an edit box. Position the box near the top of the schematic window and click to place it.
- 3 Type “**Vcc=1**” in the edit box, and then click outside of the box. A variable named “Vcc” is created.
- 4 In the Element Browser, click the **Simulation Control** category, then select the SWPVAR element and place it as shown in the following figure.



- 5 Specify the SWPVAR parameter settings shown in the following figure to define Vcc as a variable swept between 4 and 8 in steps of 0.5, then click **OK**.

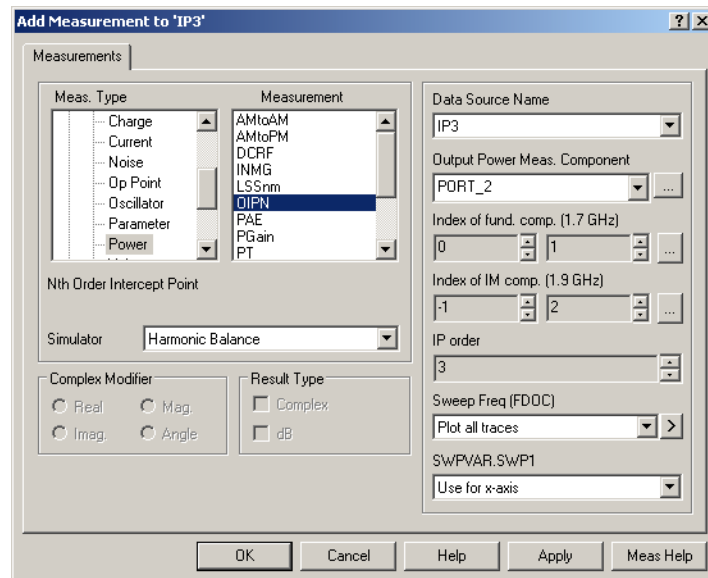




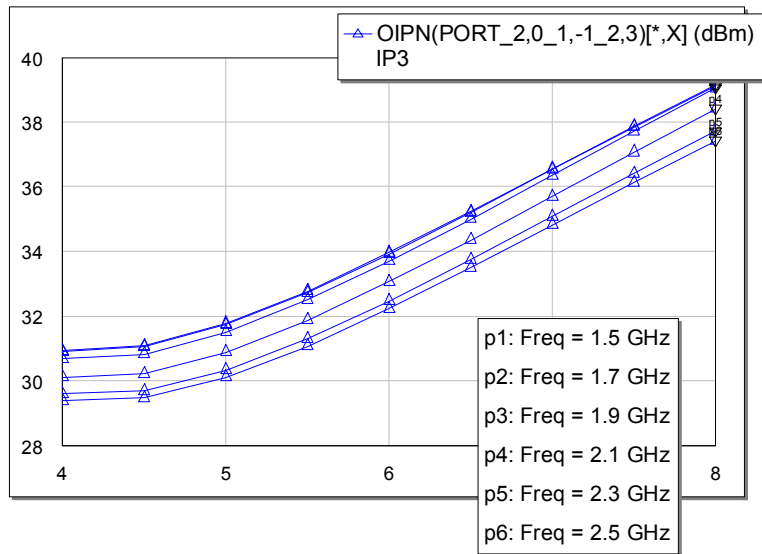
- 6 Double-click the V parameter value of DCVS V2 and change it to “**Vcc**”.

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.



- 4 Save your work by choosing **File > Save Project**.

# 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 the two approaches can be used in combination to solve many design problems. Microwave Office (MWO) supports the seamless integration of many EM simulators via an “EM Socket” software interface. To integrate a third-party simulator with the AWR Design Environment, the simulator must support an interface complementary to the “EM Socket” interface. One such simulator is the latest version of Sonnet<sup>®</sup> em<sup>®</sup> ([www.SonnetSoftware.com](http://www.SonnetSoftware.com)). Since it is not possible to document the unique features of all third-party simulators that integrate with MWO, the example presented here is centered around Applied Wave Research's native EM simulator, EMSight. The manipulation of the geometry and simulations settings in the AWR Design Environment are very similar for all simulators. Detailed documentation of the unique functionality supplied by other compatible solvers is available from the individual vendors.

EMSight can simulate planar 3D structures containing multiple metallization and dielectric layers. The structures can have interconnecting vias between layers or to ground. EMSight uses the Galerkin Method of Moments (MoM) in the spectral domain, an extremely accurate method for analyzing microstrip, stripline, and coplanar structures as well as other more arbitrary media. Properly used, this technique can provide accurate simulation results up to 100 GHz and beyond.

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

## CREATING A DISTRIBUTED INTERDIGITAL FILTER

---

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

The basic procedures in this example include:

- Creating an EM structure
- Defining an enclosure
- Adding a substrate definition
- Creating a layout
- Modelling via holes
- Viewing a structure in 3D
- Defining ports and de-embedding lines
- Viewing current density and electric fields
- Adding EM structure into schematic and simulating

### Creating a New Project

The example you create in this chapter is available in its complete form as “EM\_example.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\Microwave Office\EM* directory. You can use this example file as a reference.

To create a project:

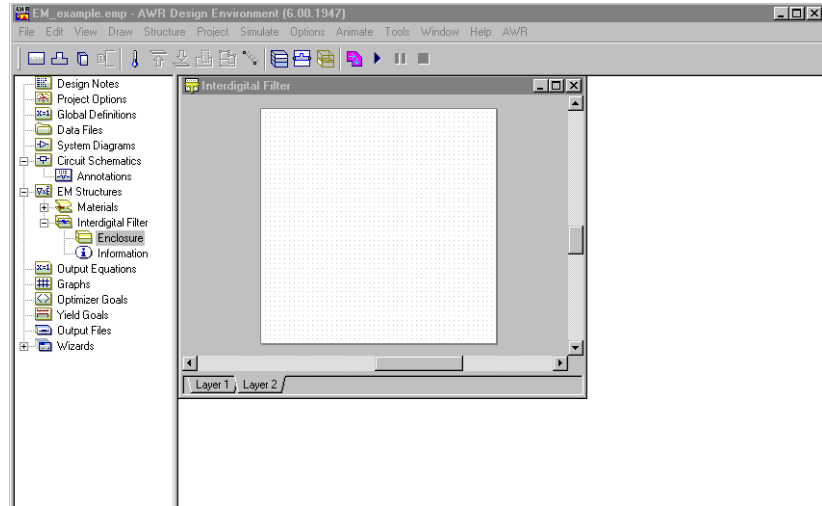
- 1 Choose **File > New Project**.
- 2 Choose **File > Save Project As**. The Save As dialog box displays.
- 3 Navigate to the directory in which you want to save the project, type “**EM\_example**” as the project name, and then click **Save**.

### 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 the AWR EMSight Simulator from the list of EM Simulators available on your computer, then click **Create**. An EM structure window displays in the workspace.



**TIP:** EMSight uses a rectilinear grid for defining structures. When you set up designs, use the coarsest grid possible when defining structures, as this provides faster simulation time (usually without any compromise in simulation accuracy.) Other third-party EM simulators may only use the grid for drawing purposes and the density may not matter-- see the third-party simulator documentation for further details.

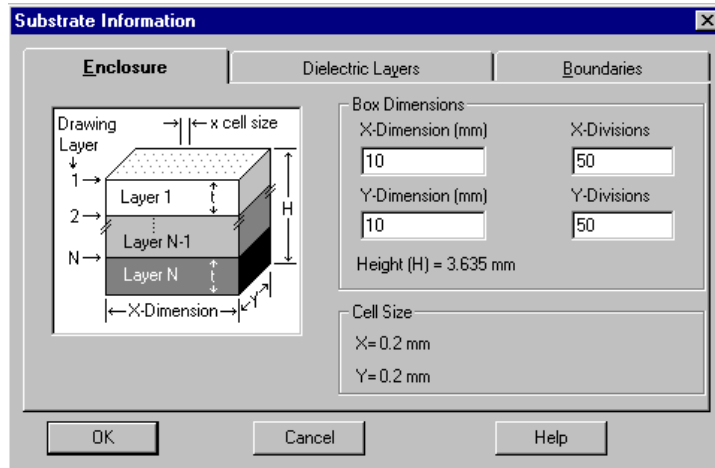
## SETTING UP THE ENCLOSURE

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

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 In the Project Browser, under **EM Structures** and “Interdigital Filter”, double-click **Enclosure**. The Substrate Information dialog box displays.

- 4 Under **Box Dimensions**, type “10” as the **X-Dimension**, type “50” in **X-Divisions**, type “10” as the **Y-Dimension**, and type “50” in **Y-Divisions**.



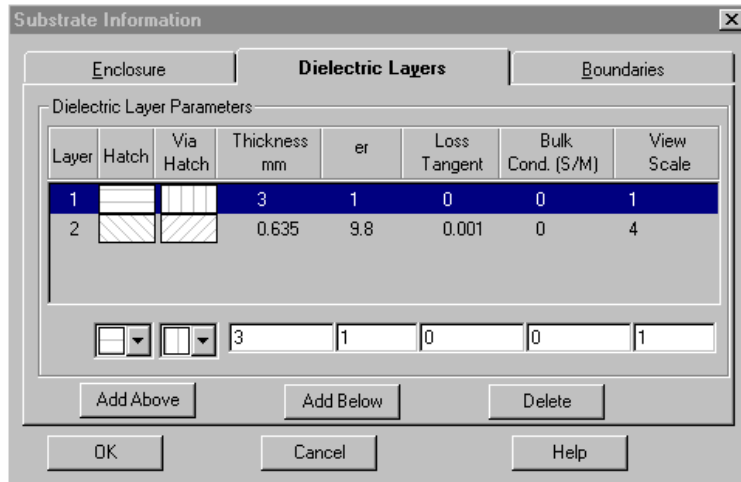
To define the dielectric layers of the enclosure:

- 5 Click the **Dielectric Layers** tab in the Substrate Information dialog box.
- 6 Select Layer 1 under **Dielectric Layer Parameters**. Type “3” in the edit box (near the bottom of the dialog box) in the **Thickness** column and type “1” in the edit box below the **er** column. Leave the default values in the remaining columns.

**TIP:**

In EMSight, simulations run twice as fast if they are loss-less. Therefore, set the Loss Tangent to zero and use perfect conductors to define all the metallization and vias in an EM structure.

- 7 Select Layer 2 under **Dielectric Layer Parameters**. Type “0.635” in the edit box in the **Thickness** column and type “9.8” in the edit box below the **er** column. Type “0.001” in the edit box below the **Loss Tangent** column and type “4” in the edit box below the **View Scale** column (this expands the 3D view for this layer to four times its normal thickness).
- 8 Select Layer 3 (if present) and click the **Delete** button.

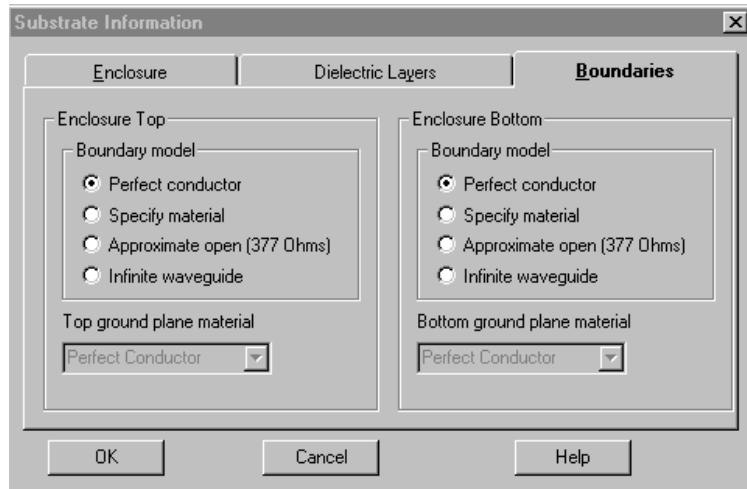


## VIEWING BOUNDARY CONDITIONS

In EMSight, the boundary conditions for the sidewalls of the enclosure are always perfect conductors and cannot be modified. 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 and bottom of the enclosure have default perfect conductors, but they can be modified. You do not modify the default boundary conditions in this example.

To view the boundary conditions:

- 1 Click the **Boundaries** tab in the Substrate Information dialog box.
- 2 Click **OK** to complete the enclosure set up procedure.



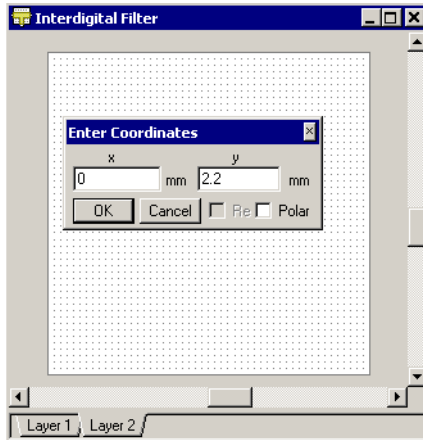
## Adding Conductors to the Layout

You can use the Microwave Office EM Simulation Editor to draw physical structures for simulation. You can also import structures directly from the Applied Wave Research 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.

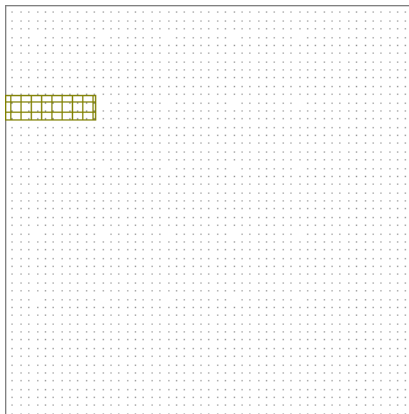
To draw the physical layout:

- 1 Ensure that the **Layer 2** tab is active in the EM structure window.
- 2 Choose **Draw > Add Rect Conductor** to add a rectangular conductor.
- 3 Move the cursor into the Interdigital Filter window, and press the **Tab** key. The Enter Coordinates dialog box displays.





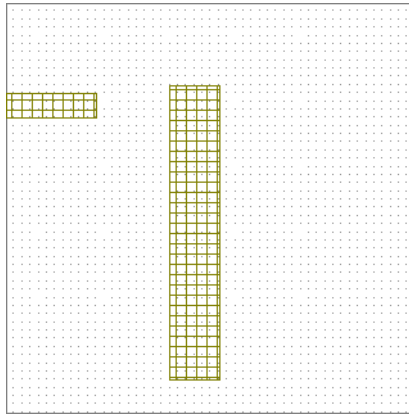
- 4 Type “0” as the value of **x** and “2.2” as the value of **y**, and then click **OK**.
- 5 Press the **Tab** key again to display the Enter Coordinates dialog box. Ensure that the **Re** (relative) check box is selected, type “2.2” as the value of **dx**, and “0.6” as the value of **dy**, and then click **OK**. A rectangular conductor displays in the EM structure window.



To draw a second rectangular conductor:

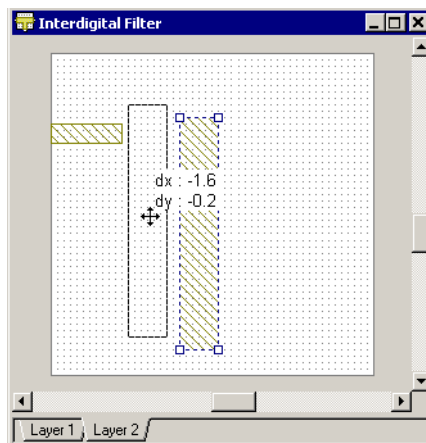
- 6 Choose **Draw > Add Rect Conductor**.
- 7 Move the cursor into the Interdigital Filter window, and press the **Tab** key. The Enter Coordinates dialog box displays. Type “4” as the value of **x** and “2” as the value of **y**, and then click **OK**.

- 8 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.



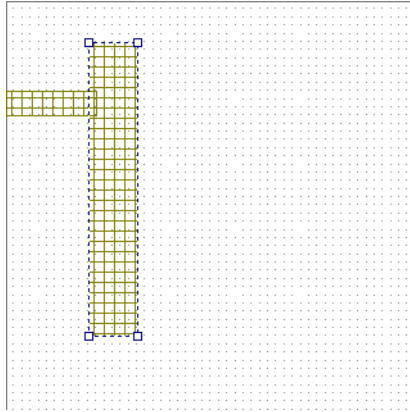
To move the second rectangular conductor next to the first conductor:

- 9 Click the second rectangular conductor. Squares display at the rectangle's corners.
- 10 Slide the cursor over the selected conductor until the cursor displays as a cross.
- 11 Click and hold down the mouse button. A dx, dy readout displays in the window, as shown in the following figure.



**TIP:** Click the **Ruler** button on the toolbar to measure the dimension of conductors, offsets, or spaces in an EM structure layout.

- 12 Drag the cursor until the dx, dy readout displays dx:-2 and dy:-1, then release the button to place the rectangle.

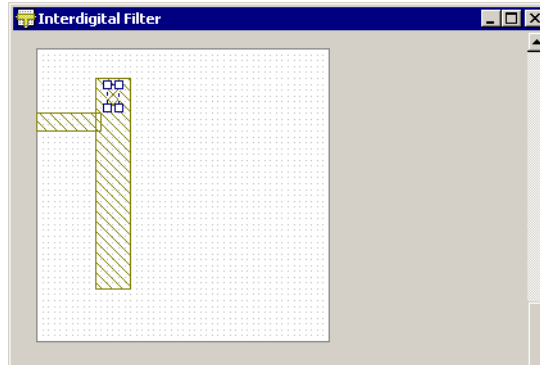


## Adding Vias

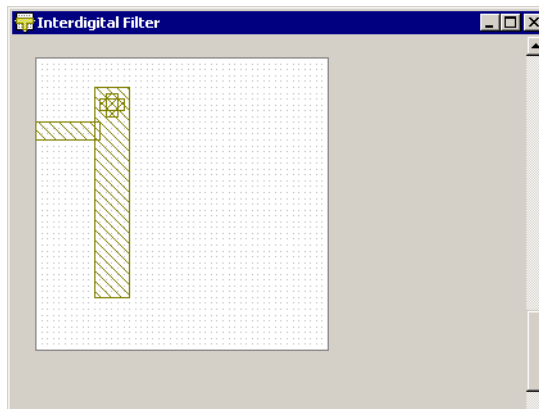
Vias are interconnects between substrate layers. You must add a via to ground one side of the larger conductor to the bottom of the enclosure.

To add a via:

- 1 Choose **Draw > Add Via**.
- 2 Move the cursor into the Interdigital Filter window and press the **Tab** key. The Enter Coordinates dialog box displays. Type “**2.4**” as the value of **x** and “**1.2**” as the value of **y**, and then click **OK**.
- 3 Press the **Tab** key again to display the Enter Coordinates dialog box. Type “**0.4**” 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, with blue squares in its corners to show that it is selected.



- 4 Choose **Edit > Copy**, then choose **Edit > Paste**.
- 5 Move the mouse into the EM structure window. An outline of the copied via displays.
- 6 Right-click once to rotate the via.
- 7 Press the **Tab** key to display the Enter Coordinates dialog box. Deselect **Re** to activate absolute coordinates. Type "**2.2**" as the **x** value and "**1.8**" as the **y** value, and then click **OK**. The new EM structure displays as follows.



## Viewing the Structure in 3D

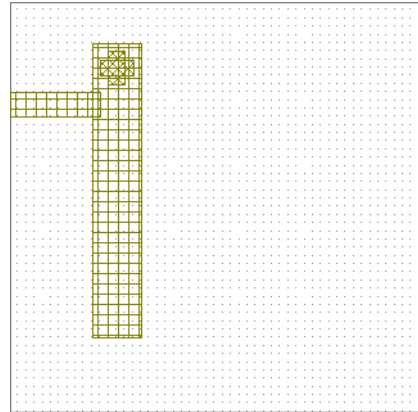
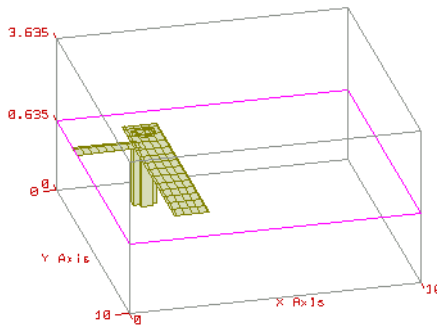
The EM simulator supports multiple 2D (structure) and 3D views.

To create a 3D view:

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

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

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



## Adding Ports and De-embedding Lines

The EM simulator can have electrical ports defined at the edge of the defining box (edge ports) or as a via probe coming in from the top or bottom surfaces (via ports).

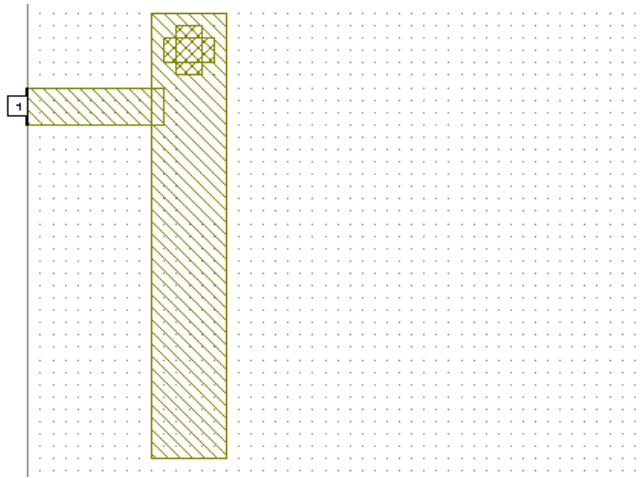
To define an edge port:

- 1 Click the smaller conductor in the EM structure window. Note that this conductor must be positioned exactly on the left edge of the substrate (X:0; Y:2.2) before you can add an edge port to it.

**TIP:** Choose **View > Zoom In** once or twice to magnify the view for the following steps.

- 2 Choose **Draw > Add Edge Port**.

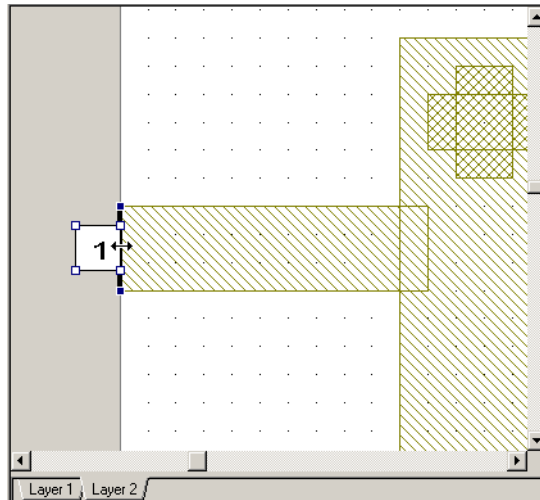
- 3 Position the cursor to the left edge of the small conductor until the outline of a square displays, and click to place the port. A small box with the number 1 (indicating port 1) displays at the left edge of the conductor.



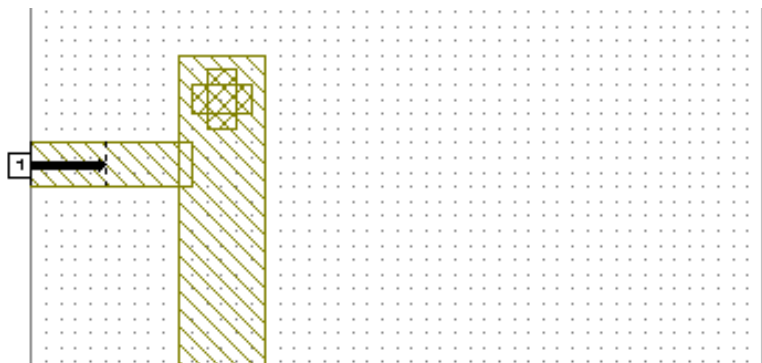
To subtract (de-embed) an amount of electrical length from the simulation results, the reference plane for the port must be moved away from the edge of the box.

To de-embed 1mm of electrical length on port 1:

- 4 Right-click in the EM structure window and choose **View Area**.
- 5 With the cursor displayed as a magnifying glass, click and drag the cursor around port 1 and the small conductor. The window zooms in on the selected area.
- 6 Click port 1. Four squares display at its corners.
- 7 Slide the mouse over the edge of the port until the cursor displays as a double arrow.



- 8 Click and hold down the mouse button to display a dx or dy readout.
- 9 Drag the cursor to the right until the dx, dy readout displays dx:1. Release the mouse button to place the de-embedding line.



## Specifying the Simulation Frequencies

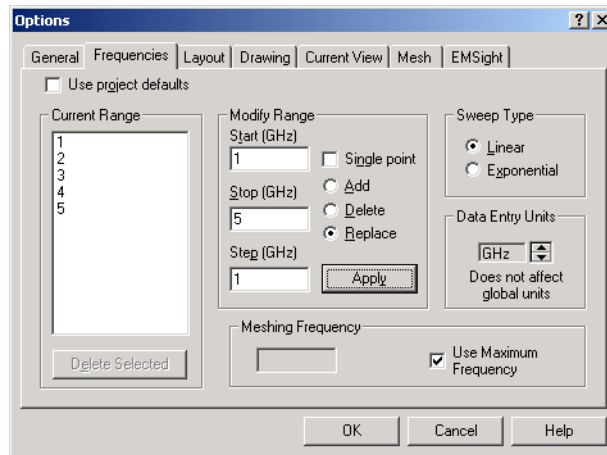
To specify the simulation frequencies:

- 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**.

**TIP:** 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 is obtained at the project frequencies using interpolation and/or extrapolation.

- 5 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**.



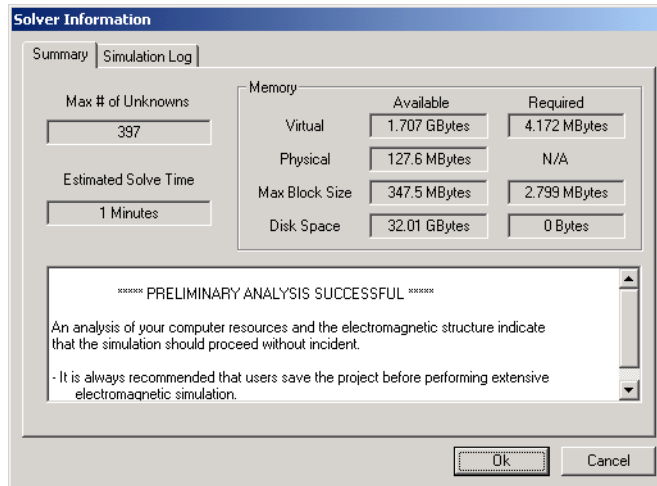
## Running the EM Simulator

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:

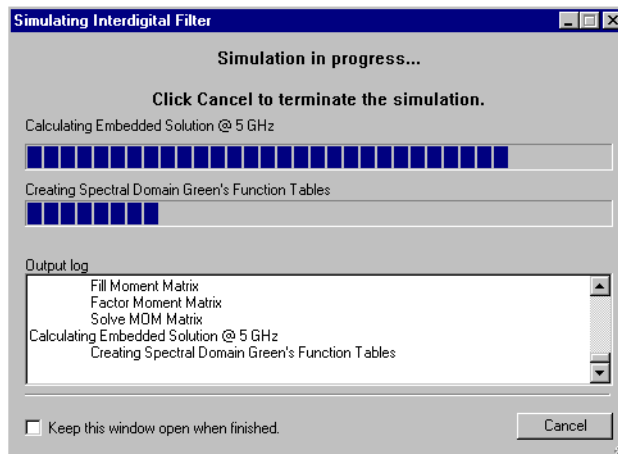
- 1 Under **EM Structures** and “Interdigital Filter”, double-click **Information**. The Solver Information dialog box displays if EMSight is the selected simulator. The estimated solve time for the EM structure and other information displays, depending on the selected solver. For more information about the data displayed for third-party simulators, see the vendor documentation.





- 2 Click **OK** to close the dialog box.
- 3 Choose **Simulate > Analyze**. A dialog displays to indicate the simulation progress. The progress bar(s) and text are controlled by the electromagnetic simulator, so the details of the steps in the analysis may vary.

**TIP:** (EMSight) AWR has unique technology that allows very large problems to be simulated even in limited computer RAM without sacrificing simulation time.

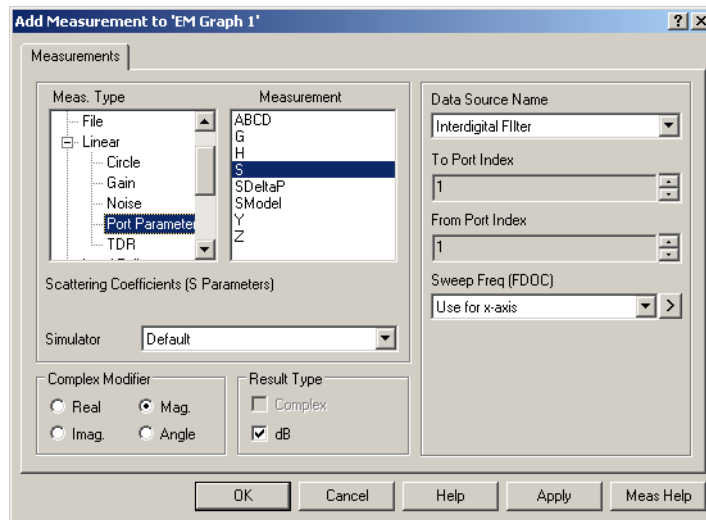


## Displaying Results on a Graph

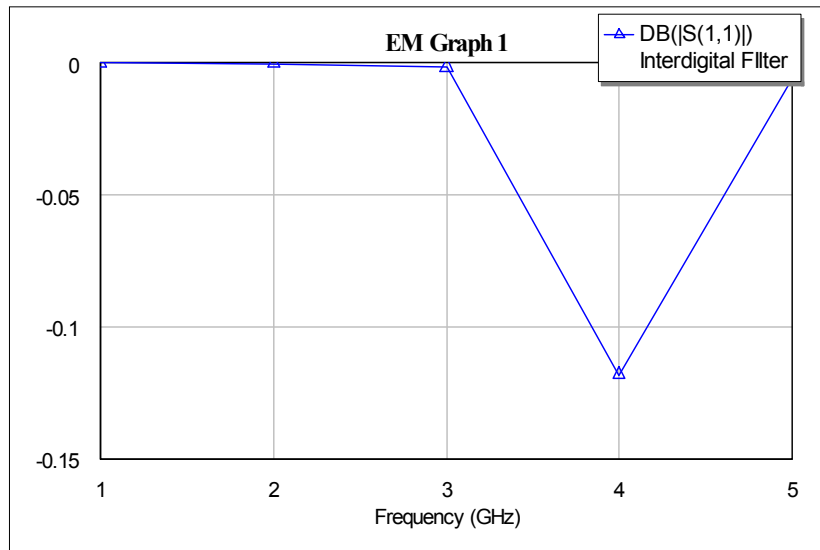
To determine the resonant frequency, you must plot the return loss of the EM structure.

To measure the resonant frequency on a graph:

- 1 In the Project Browser, right-click **Graphs** and choose **Add Graph**. The Create Graph dialog box displays.
- 2 Type “**EM Graph 1**” in **Graph Name**, select **Rectangular** as the **Graph Type**, and click **OK**. The graph displays in the workspace.
- 3 Right-click the “**EM Graph 1**” 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 near 4 GHz.

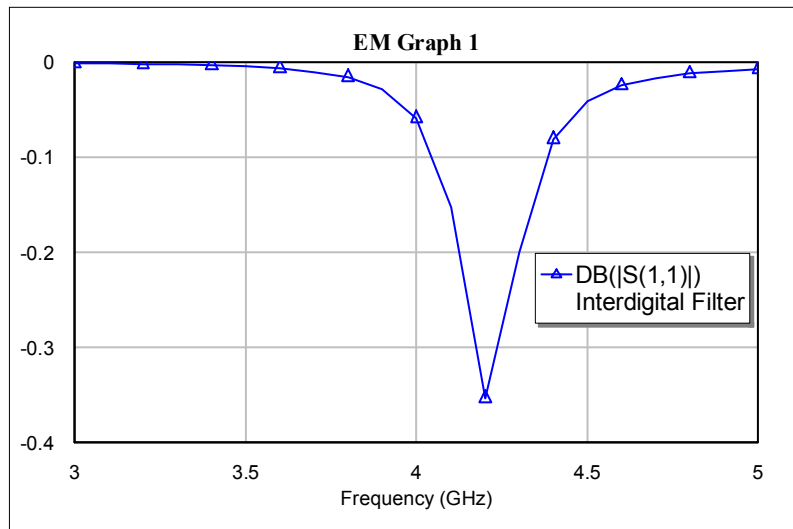


To determine a more precise measurement of the resonant frequency, you must change the frequency range and step size of the simulation.

## 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**. Click **Apply**, then click **OK**.
- 4 Choose **Simulate > Analyze** to re-analyze the circuit. The simulation response in the following graph displays.

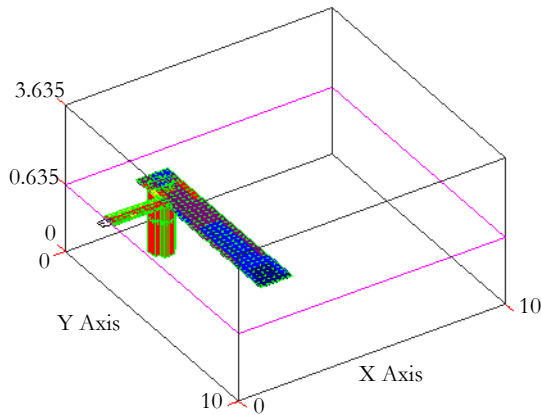


## Animating Currents and Viewing E-Fields

Viewing the currents and electric fields of an EM structure can be useful when studying its physical characteristics. Currently, EMSight is the only simulator which can display currents and E-fields within the AWR Design Environment. For details on imaging when using an alternate simulator, see the associated vendor documentation.

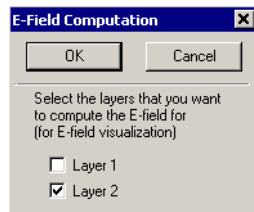
To animate the currents on the conductors:

- 1 Click the 3D window of the Interdigital Filter EM structure to make it active.
- 2 Choose **Animate > Show Current**.



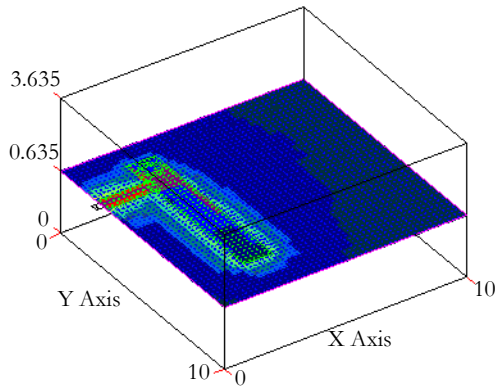
To display the electric field:

- 3 Choose **Animate > E-Field Settings**. The E-Field Computation dialog box displays.



- 4 Select the **Layer 2** check box and click **OK**.
- 5 In the Project Browser, right-click the “Interdigital Filter” under **EM Structures** and choose **Force ReSimulation**. Click **Yes** in the Force Resimulation Confirmation dialog box that displays.
- 6 Choose **Simulate > Analyze**.
- 7 Choose **Animate > Animate Play**. The animated currents in the 3D view display in the workspace.
- 8 Choose **Animate > Animate Stop** to turn off the animation.

**TIP:** This animation illustration may vary depending upon your computer configuration.

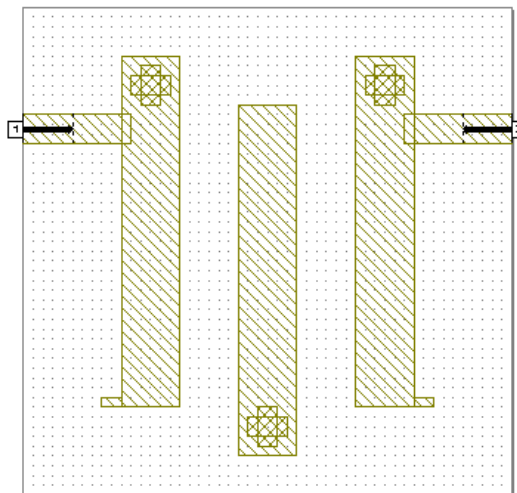


To turn off the electric field computations:

- 9 Choose **Animate > E-Field Settings**. The E-Field Computation dialog box displays.
- 10 Clear the **Layer 2** check box and click **OK**.

## Completing the Filter Layout

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



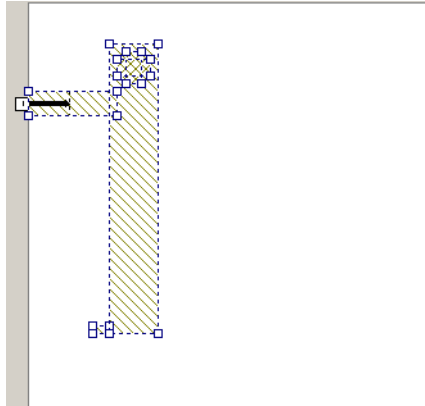
To add the small conductor on the end of the input resonator:

- 1 Click the Interdigital Filter window to make it active.
- 2 With your cursor in the window, right-click and choose **View All**.
- 3 Choose **Draw > Add Rect Conductor**. Press the **Tab** key to display the Enter Coordinates dialog box.
- 4 Type “2” as the **x** value and “8.2” as the **y** value, then click **OK**.
- 5 Press the **Tab** key again to display the Enter Coordinates dialog box. Select the **Re** check box to change the relative coordinates. 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.

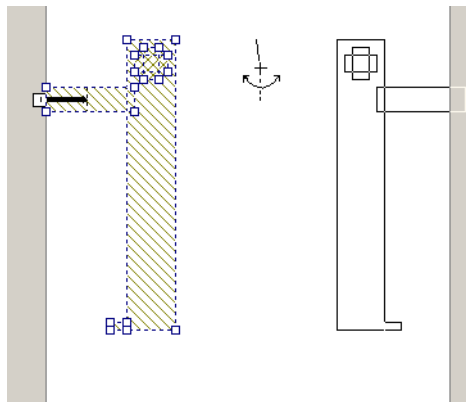


To draw the output resonator:

- 6 Choose **Edit > Select All**.
- 7 Choose **Edit > Copy**, and then choose **Edit > Paste**. An outline of the input resonator displays.
- 8 Move the cursor to place the outline of the copied instance directly on top of the original input resonator, and click. The newly created instance is still selected, as shown in the following figure.



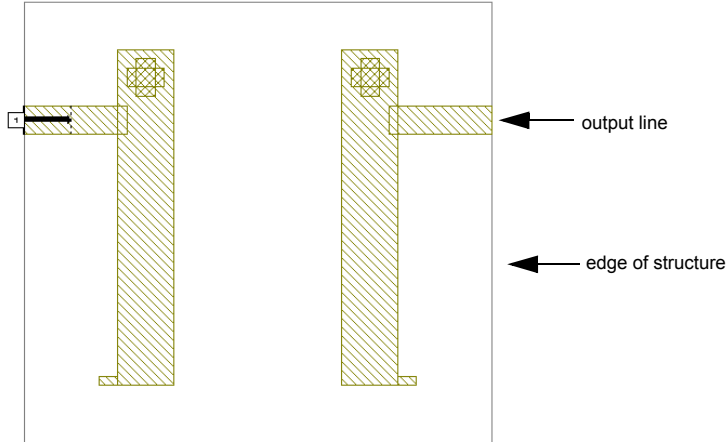
- 9 Choose **Edit > Flip**.
- 10 Move the cursor into the middle of the EM structure window. Click, and drag the cursor down, then release the mouse button. The selected instance flips as follows. Leave the flipped instance selected.



You must move the flipped instance to align the output line with the edge of the structure. To move the flipped instance:

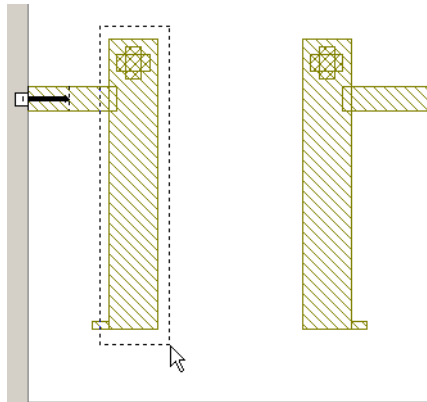
- 11 Move the cursor over the selected instance until the cursor displays as a cross.
- 12 Click, and drag the outline of the instance until the output line aligns with the edge of the structure as follows, then release the mouse button.



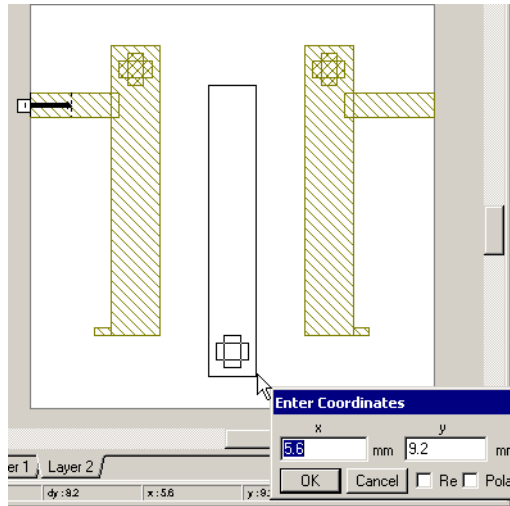


To create the middle resonator:

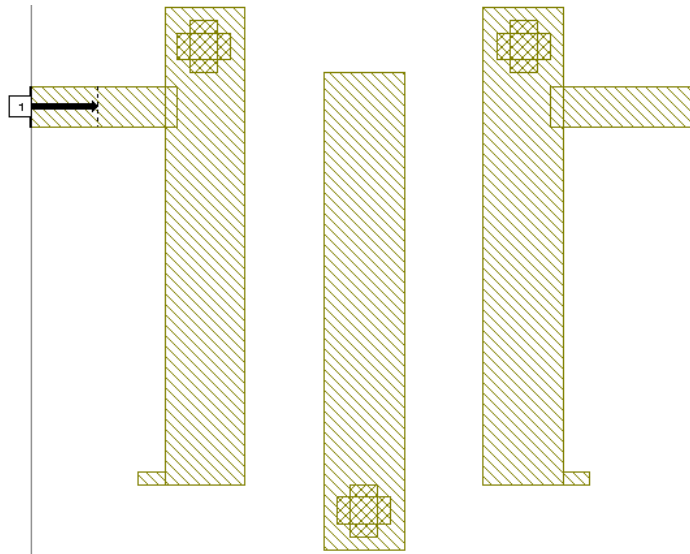
- 13 Click near the top left corner of the left-most resonator, and holding the mouse button down, drag the cursor down and to the right so the dashed box encompasses the resonator, and then release the mouse button. The large conductor and the two vias are selected.



- 14 Choose **Edit > Copy**, then choose **Edit > Paste**. An outline of the copied instance displays.
- 15 Move the cursor to the middle of the EM structure window to move the copied instance to the middle of the window, then right-click twice to rotate the instance 180-degrees.



- 16 Press the **Tab** key to display the Enter Coordinates dialog box.
- 17 Deselect **Re** to change the relative coordinates. Type “**5.6**” as the **x** value and “**9.2**” as the **y** value, and then click **OK**. The EM Structure displays as follows.



## Adding a Port

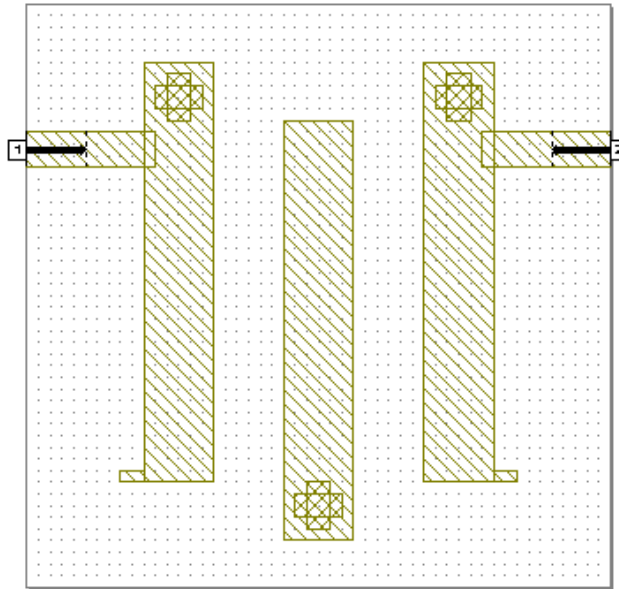
To complete the EM structure, you must add a port to the output line.

To add a port to the output line and de-embed 1mm of electrical length:

- 1 Click the rightmost conductor in the EM structure window.

**TIP:** Choose **View > Zoom In** once or twice to magnify the view for the following steps.

- 2 Choose **Draw > Add Edge Port**.
- 3 Position the cursor on the right edge of the conductor until the outline of a square displays, and click to place the port. A small box with the number 2 (indicating port 2) displays at the right edge of the conductor.
- 4 Right-click in the EM structure window and choose **View Area**.
- 5 With the cursor displayed as a magnifying glass, click and drag the cursor around port 2 and the small conductor. The window zooms in on the selected area.
- 6 Click port 2. Four squares display at its corners.
- 7 Move the cursor over the edge of the port until it displays as a double arrow.
- 8 Click and hold down the mouse button to display a dx or dy readout.
- 9 Drag the cursor to the left until the dx, dy readout displays dx:-1. Release the mouse button to place the de-embedding line. The final layout is shown in the following figure.

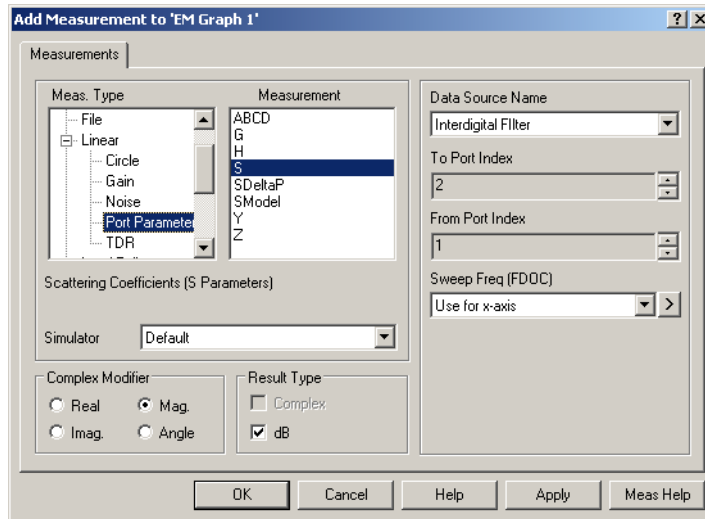


- 10 Choose **Simulate > Analyze** to re-analyze the circuit.

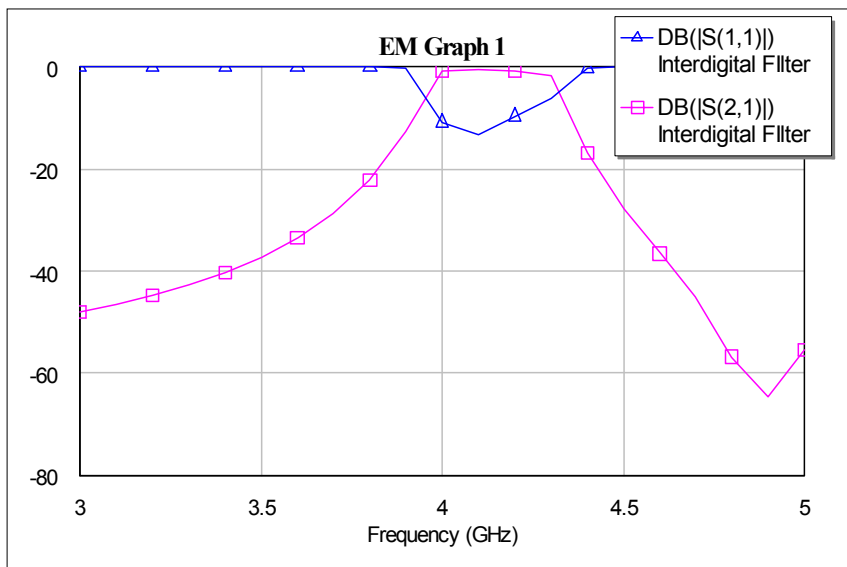
## Adding a Measurement to the Graph

To add an s21 measurement to the graph:

- 1 Right-click the “EM Graph 1” graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays.
- 2 Create a measurement using the settings in the following figure, then click **OK**.



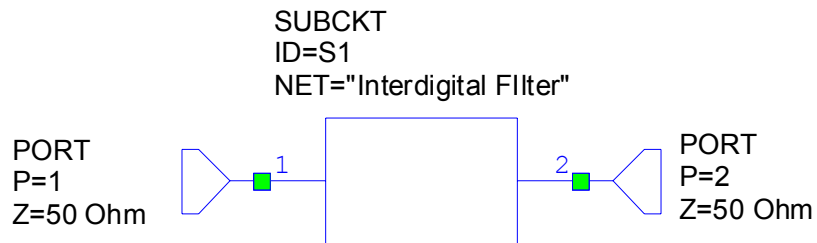
- 3 Choose **Simulate > Analyze**. The final simulation response on the following graph displays.



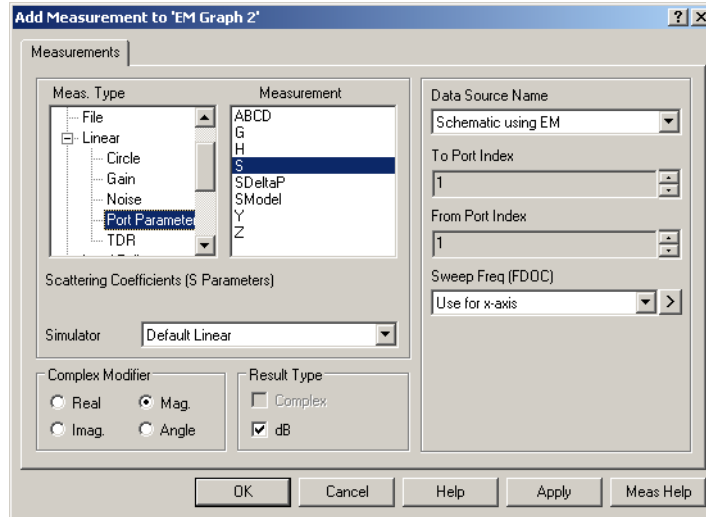
## Adding an EM Structure to Schematic and Simulating

You can add an EM structure to a schematic just as 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 (choose **Project > Add Schematic > New Schematic** or click the **New Schematic** button on the toolbar) named “Schematic using EM”.
- 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 mouse into the schematic and click to place the subcircuit.
- 4 Add two ports (click the **Port** button on the toolbar) to the subcircuit as shown in the following figure.



- 5 Right-click the “Schematic using EM ” schematic in the Project Browser and choose **Options**.
- 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 **Apply**, and then click **OK**.
- 8 Add a rectangular graph named “**EM Graph 2**”.
- 9 With the EM Graph 2 window active, click the **Add Measurement** button on the toolbar and add a measurement to the graph as shown in the following figure. Click **Apply**.



- 10 In the Add Measurement window, 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.
- 12 Save your work by choosing **File > Save Project**.





# SYSTEM SIMULATION IN VSS

---

This chapter provides a brief outline of the theory behind the Visual System Simulator (VSS), and includes a procedure for a simple amplitude modulation to demonstrate how a simulation is performed in VSS. The first section describes the basic philosophy of the simulator, and the example describes use of several key VSS features.

## OVERVIEW OF VSS THEORY

---

VSS is a sampled time-domain simulator that uses a fixed time step which is set either by the default system settings for every system diagram, or by individual blocks inside a system diagram (usually sources) and inherited by subsequent blocks. This section outlines several important aspects of a VSS simulation, including data types, the concept of Complex Envelope signal representation, center frequency and sampling frequency and their importance, and finally the concept of parameter propagation.

### Data Types

All VSS blocks have input and output nodes which handle (and operate on) data belonging to one of four basic data types: Digital, Real, Complex, or Unset. Each VSS block node color corresponds to its data type: green for Digital, yellow for Real, red for Complex, and white for an Unset data type. Unset nodes indicate the block supports two or more data types. You can double-click an unset node to redefine it as a specific node type. For example, ADD, an  $n$ -input adder (located in the Element Browser under **System Blocks** in the **Math Tools > Adders** category) has Unset nodes by default, signifying that it adds the data coming into its nodes and provides the sum at its output node regardless of the data type. Another example is the behavioral amplifier AMP\_B (located in the Element Browser under **System Blocks** in the **RF Blocks > Amplifiers** category), which also has its ports unset. The amplifier block supports both real and complex signals, but does not support digital signals.

Digital data types comprise streams of digital data with abrupt transitions (such as a pseudo-random sequence of bits generated by a source to perform a

Monte-Carlo simulation of a digital communication system). Real data refers to any real waveform observed in communication systems, for example, sinusoids, real passband noise, or possibly a sawtooth waveform. You can use these two data types to represent any waveform encountered in natural system design. Complex data deserves more attention because it is a compact way to represent complex baseband data (with frequency content concentrated around DC, as required by modern communication systems), as well as real passband waveforms via the Complex Envelope (CE) signal representation.

## Complex Envelope Signal Representation

As a sampled time-domain system simulator, a basic underlying concept of VSS is that of sampling frequency-- something not encountered in nature (where all signals and waveforms are analog, i.e., they exist in continuous time), but more and more prevalent in modern measurement equipment, a growing portion of which samples the measured data at a uniform specified rate before manipulating or processing it.

In general, sampling an analog signal into a discrete time representation is an information-lossless operation only if the sampling frequency exceeds two times the highest frequency content of the analog signal. In such cases, recovery of the original analog waveform from its sampled stream is typically perfect via an ideal lowpass filter. Otherwise, a phenomenon known as aliasing occurs, and it is not possible to reconstruct the original analog waveform based on the sampled stream.

This concept places a significant burden on any time-domain systems simulator, because in a simulated transmitter/receiver chain it forces the overall sampling frequency to be at least twice the highest frequency component anywhere in the system. This is somewhat wasteful, since up- or down-conversion chains usually include carrier-modulated passband signals of relatively low frequency content (bandwidth) concentrated around a very high frequency carrier. In principle, the signal of interest is the modulating signal and not the carrier, and assuming an interest in a narrow band of frequencies around a center frequency, this modulating signal can be more efficiently represented by its Complex Envelope (CE), assuming the frequencies very far away from the carrier are filtered out anyway. For example, consider a GSM signal, which has a bandwidth of a few hundred kHz, but is modulated on a 1.9 GHz carrier. In principle, a sampling frequency of 5 MHz (more correctly, 5 Msamples/sec) can adequately describe the signal in its CE form, but to also comfortably sample the carrier, the sampling frequency must be at least 3.8 GHz, and more comfortably 5 GHz, or

5 Gsamples/sec. It is obvious how the two different approaches can result in a simulation speed difference of three orders of magnitude!

VSS utilizes the CE representation of signals whenever possible to gain the tremendous advantage in simulation speed discussed here, without compromising simulation accuracy. Specifically, a real passband signal  $x(t)$ , representing a narrowband modulation centered about a high frequency sinusoidal carrier with frequency  $f_c$  is mathematically represented as:

$$x(t) = x_c(t) \cdot \cos 2\pi f_c t - x_s(t) \cdot \sin 2\pi f_c t,$$

where  $x_c(t)$  and  $x_s(t)$  are real lowpass signals, with bandwidth much smaller than the carrier frequency  $f_c$ , and are called the in-phase and quadrature components of the real passband signal  $x(t)$ . The signal can be represented by its Complex Envelope (CE) form  $c(t)$ , where:

$$x(t) = \text{Re}\{c(t) \cdot e^{j2\pi f_c t}\},$$

and, therefore, it holds that the CE lowpass signal is:

$$c(t) = x_c(t) + j \cdot x_s(t).$$

VSS utilizes the CE lowpass equivalent signal  $c(t)$  wherever possible to allow for orders-of-magnitude-faster narrowband simulation. To this end, each signal at any point in the simulation has a sampling frequency, and a *center frequency tag* associated with it. For example, a plain tone at frequency 2 GHz for which the real passband signal is  $x(t) = \cos 2\pi f_c t$ , can be easily generated using the SINE block (located under **System Blocks** in the **Sources > Waveforms** category) with its output node set to complex and represented in CE form:

- by leaving the center frequency (CTRFRQ) empty and setting frequency (FRQ) to 2GHz resulting in  $c(t) = 1.0 + j \cdot 0.0$  bearing a center frequency tag of 2 GHz, or
- by having a center frequency (CTRFRQ) of, for example, 1 GHz and frequency (FRQ) of 5 GHz, in which case  $c(t) = \exp(j2\pi \cdot (FRQ - CTRFRQ)t)$  bears a center frequency tag of 1 GHz, or
- by having a center frequency (CTRFRQ) of 0 and a FRQ of 2 GHz, in which case  $c(t) = \exp(j2\pi \cdot 2e9 \cdot t)$  bears a center frequency tag of 0.

When working with RF tones, the TONE block, located under **System Blocks** in the **RF Blocks > Sources** category, is preferred, as all frequencies are specified in absolute frequency and power is specified in dB/dBm.

All of these CE forms show the same spectrum plot (the 2 GHz tone corresponding to the real passband signal  $x(t) = \cos 2\pi f_c t$ ), although the time

domain waveform generated by VSS is internally different. The center frequency tag is a parameter propagated implicitly, but internally the signal is modeled as a CE lowpass equivalent.

As another example, the GSM signal previously discussed would also be in CE lowpass equivalent form in VSS; a sequence of complex numbers sampled at 5 MHz and bearing only a center frequency tag of 1.9 GHz, and not a series of real samples taken at a rate of 5 Gsamples/sec. Of course, if the latter more cumbersome approach is desired, VSS provides the capability to switch any signal to real passband representation via the CE-to-Real block (CE2R) located under **System Blocks** in the **Converters > Complex Envelope** category.

Note that VSS treats complex signals depending on their context, their center frequency, and the block performing the operation. For example, blocks found under the System Blocks **Math Tools > Math Functions** category simply perform standard complex arithmetic on their input complex signals, treating them as ordinary complex numbers. Modulation mapper and detection blocks in the **Modulation** category treat the series of complex samples as baseband I/Q symbols. Blocks designed to operate on RF signals, such as those in the **Filters** or **RF Blocks** categories treat complex signals with non-zero center frequency as CE representations of a real signal centered around a carrier at the center frequency. When the center frequency is 0, by default the RF amplifier, RF mixer and circuit filter blocks treat the complex signal as a pair of real signals representing separate I and Q channels.

## Center Frequency and Sampling Frequency

It is important to note that the CE representation of signals can greatly reduce simulation time, but requires careful choice of the sampling frequency, such that the frequencies of interest are included in a simulation. Any simulated CE signal only exists for frequencies that lie in the interval:

$$\left[ f_c - \frac{f_s}{2}, f_c + \frac{f_s}{2} \right],$$

where  $f_c$  is the center frequency of the signal, and  $f_s$  is the sampling frequency.

Therefore, to examine the frequency content (for example, the Adjacent Channel Power Ratio, or ACPR) of the previous GSM signal at a frequency offset of 30 MHz from the 1.9 GHz carrier (the signal's center frequency tag), you must make sure the sampling frequency is at least  $f_s \geq 60$  MHz, so that the signal exists between 1.87 GHz and 1.93 GHz, or  $[f_c - (f_s/2), f_c + (f_s/2)]$ .

Because VSS is geared towards digital communication applications, many of its blocks (and the entire system diagram) have Data Rate and Oversampling associated with them.

The Data Rate is the number of digital communication symbols per second. Inside a VSS system diagram the default data rate is denoted `_DRATE`. A symbol can differ in meaning, depending on the modulation specifics. For example, for the previous GSM, the symbol rate or data rate is set by the standard as 270.833 ksymbols/sec, and since in this case every symbol is one bit, it translates to 270.833 kbits/sec. To simulate a satellite link using Quadrature Phase Shift Keying (QPSK) modulation to transmit 100 Mbits/sec, you set the symbol rate (or data rate) of the QPSK source block to 50 Msymbols/sec (because each QPSK symbol corresponds to 2 bits). The QPSK\_SRC block is found under the System Blocks **Modulation > QPSK** category.

Each of these symbols can be represented with any number of samples (oversampling). Inside a VSS system diagram the default number of samples per symbol is denoted `_SMPSYM`. For the QPSK example, you can have 10 samples per symbol, which is a total sampling frequency of:

$$f_s = (\text{DataRate}) \cdot (\text{Oversampling}) = 500 \text{ MHz.}$$

As previously explained, if the center frequency tag of this QPSK signal is 5 GHz, the signal will exist for 250 MHz on either side of the 5 GHz carrier (from 4.75 GHz to 5.25 GHz).

For digital communications, the data rate and oversampling values, and the center frequency tag of each signal are important. You can set these values on the **Simulator** tab of the System Simulator Options dialog box (as shown in the following example) or in the source blocks in the simulation (usually at the beginning of a simulated chain) and they are subsequently propagated along any constructed simulation chain. At any point in the system diagram you can use the System Tools FRQ\_PROP measurement to check the propagated parameters.

Most of the blocks that have either a `DRATE` or `SMPFRQ` parameter have a default value of empty for these parameters. When the value is empty, the blocks will automatically determine their data rate or sampling frequency. If a downstream block somehow specifies the sampling frequency, either directly or due to other blocks connected to it, that value is used. Otherwise, the rate is determined from the default settings from the Options dialog box of a system diagram.

## Parameter Propagation

An important VSS feature for increasing ease of use is *Parameter propagation*, introduced briefly when previously discussing propagation of the sampling frequency and the center frequency by all VSS blocks to other blocks further downstream in the simulation chain. This procedure of parameter propagation is bidirectional, and also occurs from the end to the beginning of a simulation chain. In VSS, the forward and backward parameter propagation occurs for a variety of parameters, a small set of which are center frequency, sampling frequency, oversampling, signal and noise levels, and delay and phase distortion.

For example, you can place a QPSK transmitter inside a system diagram, configure it for the properties of the specific transmission scenario (data rate, pulse shaping, power, etc.), and not repeat the corresponding settings in a receiver block. This is done automatically via parameter propagation by the simulator at the start-up phase of each simulation. Even more impressively, you can place an amplifier block and/or a filter somewhere in the simulation chain between the transmitter and receiver, and then *not* need to adjust the signal arriving at the receiver for delay and phase rotation introduced by the filter, or for gain introduced by the amplifier. All of these parameters are automatically propagated forward by the simulator, thus allowing the receiver block to adjust the received signal for them. As a result, even the first time, you can set up and run a relatively involved BER simulation of a transmitter/receiver chain in just a few minutes.

The details of parameter propagation for each individual block are explained in the block Help. For instance, an amplifier doesn't alter the propagated value of the center frequency tag at its input, but does alter the propagated signal and noise levels, according to its gain (and possibly noise figure). A mixer block with a center frequency  $f_{m,c}$  arriving at its input node, and a center frequency  $f_{LO,c}$  arriving at its LO node propagates as a center frequency either the sum  $f_{m,c} + f_{LO,c}$  (if it is in up-conversion mode) or the difference  $|f_{m,c} - f_{LO,c}|$  (if it is in down-conversion mode). A filter block increases the propagated value of delay at its output by adding to the propagated delay at its input the amount of delay it introduces itself to the signal.

## AM-MODULATION EXAMPLE

---

In this example a sinusoidal data signal with a frequency of 2 GHz is modulated onto a sinusoid carrier of 40 GHz.

AM modulation is described as:

$$X_{AM}(t) = C \cdot [A + m(t)] \cos \omega_c \cdot t$$

where  $m(t)$  is the message data signal; a sinusoidal signal of frequency 2 GHz given by:

$$m(t) = B \cos \omega \cdot t$$

$A$  represents the DC level of the message signal and  $B$  and  $C$  represent the amplitudes of the carrier and the message signal respectively.

The procedures in this example include:

- Creating a project
- Setting default system settings
- Creating a system diagram
- Placing blocks in the system diagram
- Specifying System Simulator options
- Adding graphs and measurements
- Running the simulation and analyzing the results

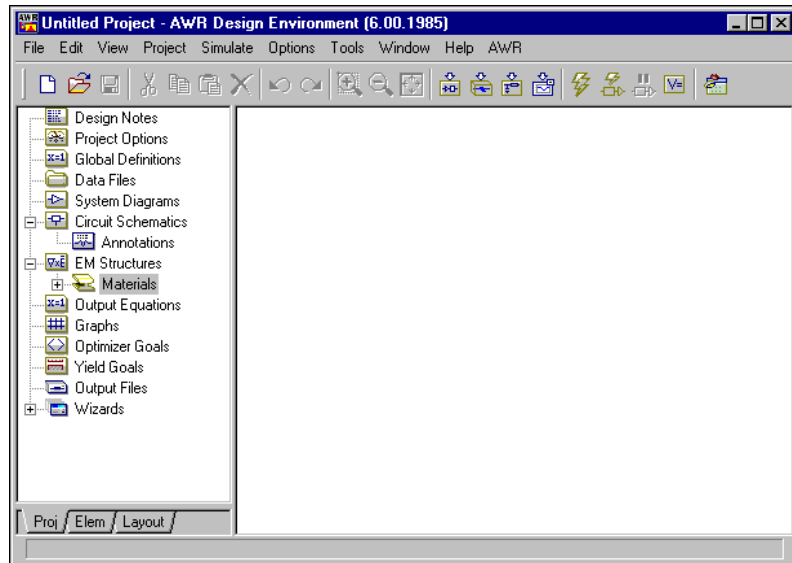
## Creating a Project

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

The example you create in this chapter is available in its complete form as “AM.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\AM* directory. You can use this example file as a reference.

To create a project:

- 1 Start VSS if not already started. To start VSS, click **Start** on your desktop, choose **Programs > AWR Suite 2004 > AWR Design Environment**, or double-click the corresponding shortcut on your desktop. For information on installing, setting up shortcuts and starting VSS, see “Installing MWO/ VSS” on page 2-1.



- 2 Choose **File > New Project**.
- 3 Choose **File > Save Project As**. The Save As dialog box displays.
- 4 Navigate to the directory in which you want to save the project, type “**AM**” as the project name, and then click **Save**. The project name displays in the title bar.

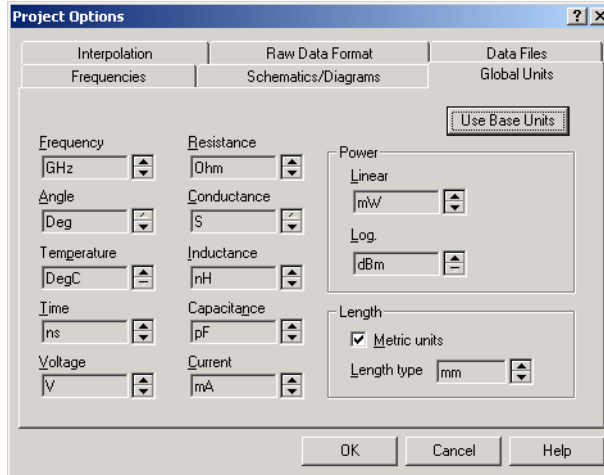
## Setting Default System Settings

Before creating a simulation you should set the default system settings.

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.





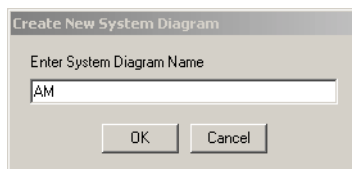
- 3 Click **OK** to save your settings.

## Creating a System Diagram

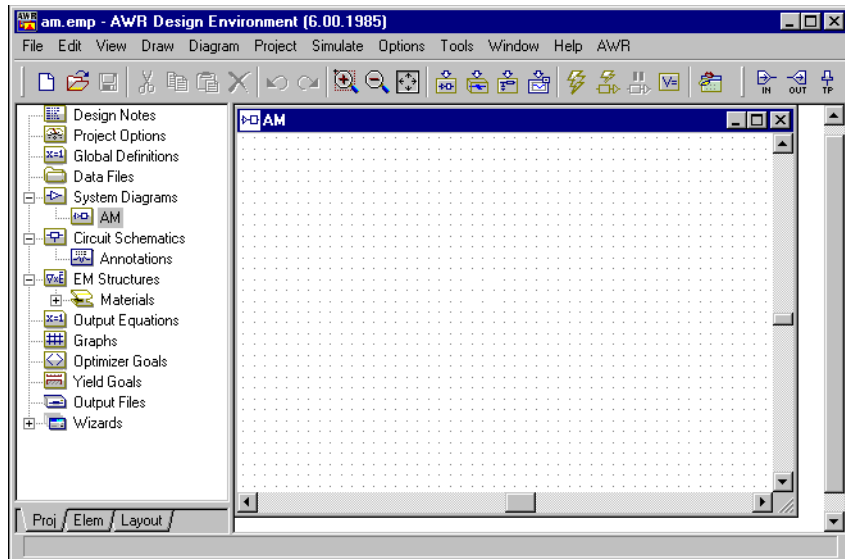
The system diagram is the canvas upon which you build end-to-end communications systems and graphically develop algorithms using VSS behavioral blocks. A VSS project can include multiple system diagrams, linear and nonlinear schematics, and netlists.

To create a system diagram:

- 1 Choose **Project > Add System Diagram > New System Diagram**. The Create New System Diagram dialog box displays.



- 2 Type **"AM"**, and click **OK**. A system diagram window displays in the workspace and the "AM" system diagram displays under **System Diagrams** in the Project Browser.



## Placing Blocks in a System Diagram

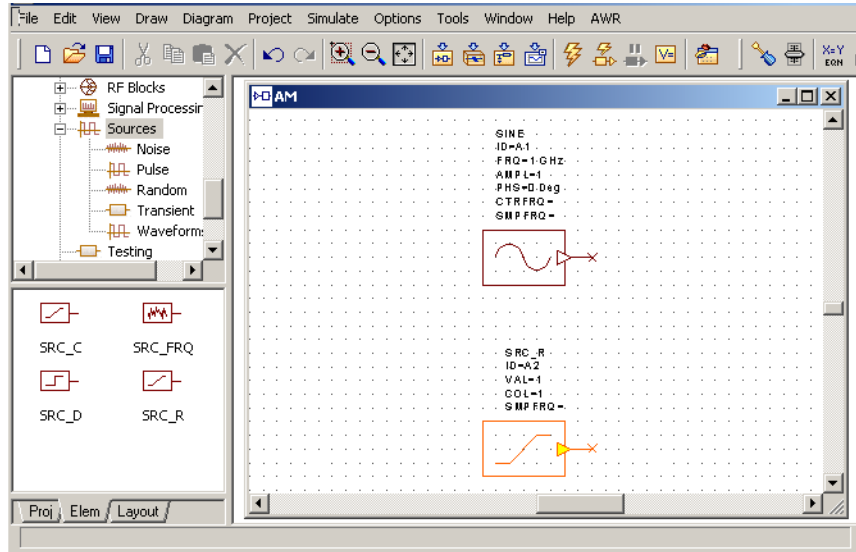
The Element Catalog is a database of behavioral system blocks that you can include in your system diagrams.

To place a system block in a system diagram:

- 1 Click the **Elem** tab in the lower left of the window to display the Element Browser. The Element Browser replaces the Project Browser window.
- 2 If necessary, click the + symbol to the left of the **System Blocks** node to expand the system blocks tree.
- 3 Click the **Sources** category. A Real Source block (SRC\_R) displays in the lower pane.
- 4 Select the SRC\_R block and drag it onto the system diagram, release the button, and then click to position the element as shown in the following figure. This serves as the DC level of the message signal.

**TIP:** You can view the full name of a system block before dragging it to the system diagram by simply moving the mouse over the block or right-clicking on it and then selecting **Show Details**.

- 5 Expand the **Sources** category, then click the **Waveforms** subgroup. Select the SINE block and place it as shown in the following figure.



**TIP:** Before clicking to position a block, you can rotate the block by right-clicking on it.

- 6 Expand the **Math Tools** category, then click the **Adders** subgroup. Select the ADD block and place it as shown in the following figure.
- 7 Expand the **Modulation** category, then click the **Analog** subgroup. Select the AM\_MOD block and place it as shown in the following figure.
- 8 Select the SINE block in the system diagram. Choose **Edit > Copy** then **Edit > Paste**. Place the duplicated block as shown in the following figure.

**TIP:** Choose **View > Zoom In** to magnify the system diagram.

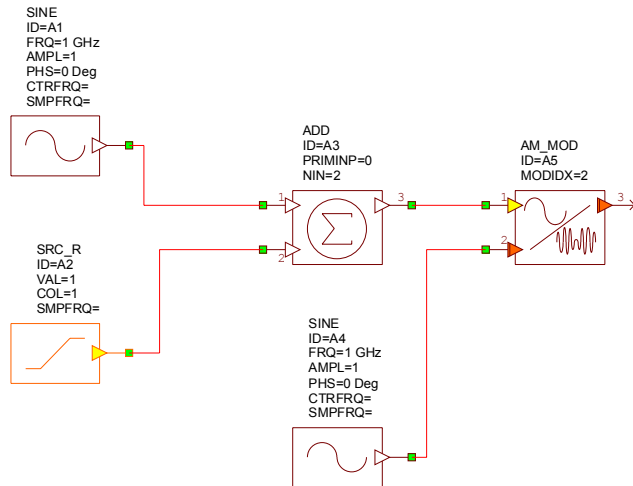
- 9 To save the file choose **File > Save Project**.

## CONNECTING THE BLOCKS AND ADDING TEST POINTS

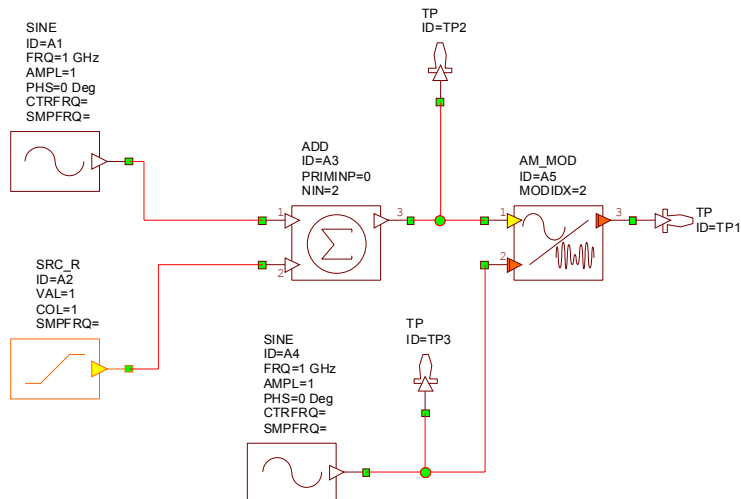
To connect the system blocks and add Test Points:



- 1 Place the cursor over the node of the SRC\_R block. The cursor displays as a wire coil symbol.
- 2 Click and drag the displayed wire to input node 2 of the ADD block, then click to place the wire.
- 3 Repeat steps 1 and 2 to complete the connections shown in the following figure.



- 4 From the system block list click the **Meters** category. Individually select three Test Points (TP) and place them as shown in the following figure. You can also click the **Add Test Point** button on the toolbar. While placing the test points, right-click to rotate them as needed. The simulation results can be displayed at these test points.

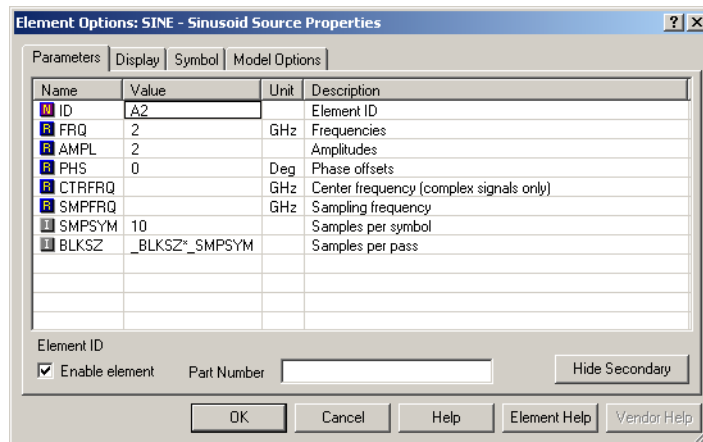


**TIP:** You can also connect blocks by moving them to snap their nodes together. When they are properly connected a small green square displays and the connection wire extends if you move either block. If you do not see the green square, try to drag one of the block into place again.

## EDITING BLOCK PARAMETERS

To edit block parameters:

- 1 In the system diagram, double-click the SINE block connected to the ADD block. The Element Option dialog box displays.
- 2 Click **Show Secondary** to display the secondary parameters. Edit the parameters to the values shown in the following figure, then click **OK**.



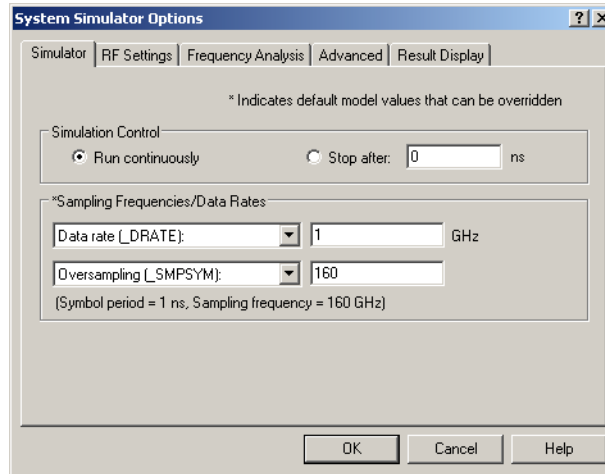
- 3 Double-click the SINE block connected to the AM\_MOD block. If the secondary parameters are not visible, click **Show Secondary**. Change the FRQ parameter to “**40**”, the AMPL parameter to “**3**”, the CTRFRQ parameter to “**0**”, and the SMPSYM parameter to “**10**”, then click **OK**.
- 4 Double-click the SRC\_R block and change the SMPSYM parameter to “**10**”, then click **OK**.
- 5 Double-click the AM\_MOD block and change the MODIDX parameter to “**2**”, then click **OK**.

**TIP:** You can also simply double-click the parameter value displayed on the system diagram to modify a single parameter.

## Specifying System Simulator Options

To specify system simulation sampling:

- 1 Choose **Options > Default System Options**. The System Simulator Options dialog box displays.



- 2 Click the **Simulator** tab, and under **Sampling Frequencies/Data Rates** select **Data rate** in the first drop-down text box, and **Oversampling** in the second drop-down text box.
- 3 Type “1” GHz as the **Data rate** and “160” as the **Oversampling** value, then click **OK**. Note the options under **Sampling Frequencies/Data Rates** that define the system sampling rate. Data rate, symbol period, time step, sampling frequency and oversampling rate are all interrelated. You can use various pairs of these settings to specify the overall settings.

## Creating a Graph to View Results

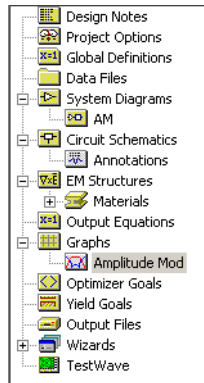
VSS allows you to see the results of your simulations in various graphical formats. Before you perform a simulation, you must create a graph and specify the data or measurements that you want to plot.

To create a graph:

- 1 Click the **Proj** tab in the lower left the window to display the Project Browser.



- 2 Right-click **Graphs** and choose **Add Graph**. You can also click the **New Graph** button on the toolbar. The Create Graph dialog box displays.
- 3 Type “**Amplitude Mod**” in **Graph Name**, select **Rectangular** as the **Graph Type**, and click **OK**. The graph displays in a window in the workspace and displays as a subgroup of **Graphs** in the Project Browser.

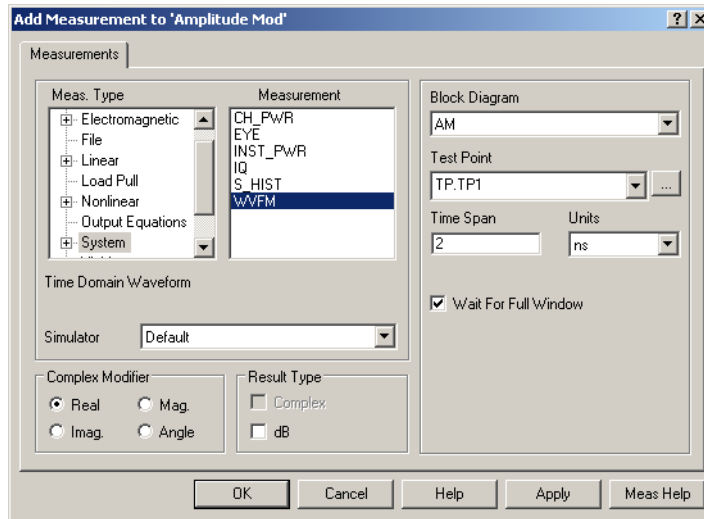


## Adding a Measurement

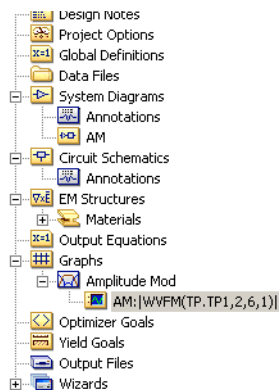
To add a measurement to the graph:



- 1 Right-click the “Amplitude Mod” graph in the Project Browser, and choose **Add Measurement**. The Add Measurement dialog box displays. You can also click the **Add Measurement** button on the toolbar.
- 2 For measurement type, select **System** under **Meas. Type** and select **WVFM** under **Measurement**.
- 3 Type “**2**” as the **Time Span** and select **ns** as the **Units** and select **Real** as **Complex Modifier**.



- 4 Ensure that **Test Point** is **TP.TP1**, then click **Apply**. The  $AM:Re(WVFM(TP.TP1,2,6,1))$  measurement displays under the “Amplitude Mod” graph in the Project Browser.



- 5 Select TP:TP2 in **Test Point**, then click **Apply**.
- 6 Select TP:TP3 in **Test Point**, then click **OK**.

**TIP:** You can custom name a test point by double-clicking its ID number.

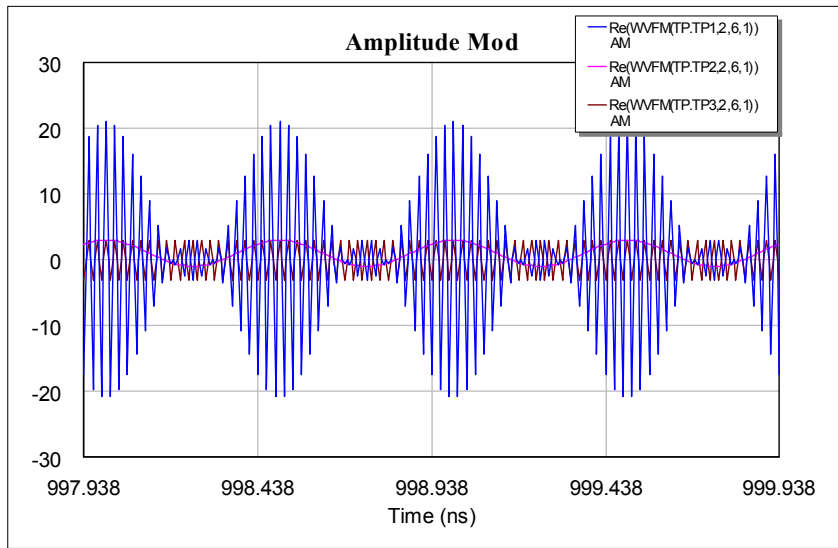


## Running the Simulation and Analyzing Results



To run the simulation:

- 1 Choose **Simulate > Run/Stop System Simulators**. Let the simulation run for 5 seconds, then choose **Simulate > Run/Stop System Simulators** again to stop the simulation. You can also click the **Run/Stop System Simulators** button on the toolbar. The simulation response in the following graph should display.



- 2 Choose **File > Save Project** to save the project.
- 3 Choose **File > Close Project** to close the project.



## END-TO-END SYSTEM

This chapter illustrates the signal and noise power relationship in an end-to-end communication link system. The goal of an end-to-end link analysis is to measure how often a transmitted bit is received in error (BER). Sometimes it is preferable to deal with symbols (a bit or group of bits encoded in a signal). In this example you evaluate the link error rate for a basic QAM transmission. You also analyze how often bits (BER) or symbols are received in error (SER), and the effect of signal-to-noise (SNR) on BER and SER.

The procedures in this example include:

- Creating a QAM project and system diagram
- Creating graphs and analyzing BER and SER
- Tuning the system parameters.

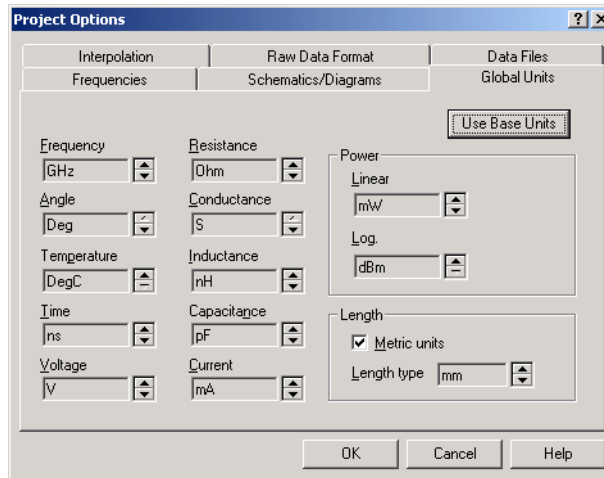
## CREATING A QAM PROJECT

---

The example you create in this chapter is available in its complete form as “QAM.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory. 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 “QAM” as the project name, and then click **Save**.
- 4 Choose **Options > Project Options**.
- 5 Click the **Global Units** tab and verify that your settings match those in the following figure.



- 6 Click **OK** to save your settings.

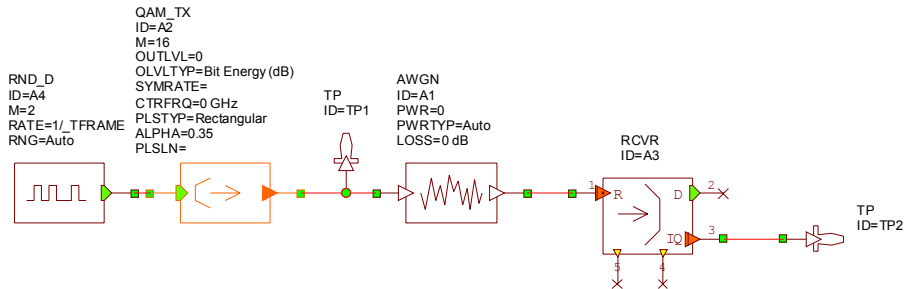
## Creating a QAM End-to-End System Diagram

To create a QAM end-to-end system diagram:

- 1 Choose **Project > Add System Diagram > New System Diagram**. The Create New System Diagram dialog box displays.
- 2 Type “**QAM**” as the diagram name and click **OK**.
- 3 Click the **Elem** tab in the lower left of the window to display the Element Browser.
- 4 Expand the **Sources** category, then click the **Random** subgroup. Select the RND\_D block and place it on the system diagram as shown in the following figure.
- 5 Expand the **Modulation** category, then click the **QAM** subgroup. Select the QAM\_TX block and place it on the system diagram as shown in the following figure.
- 6 Click the **Channels** category, then select the AWGN block and place it on the system diagram as shown in the following figure.
- 7 In the **Modulation** category, click the **General Receivers** subgroup. Select the RCVR block and place it on the system diagram as shown in the following figure.

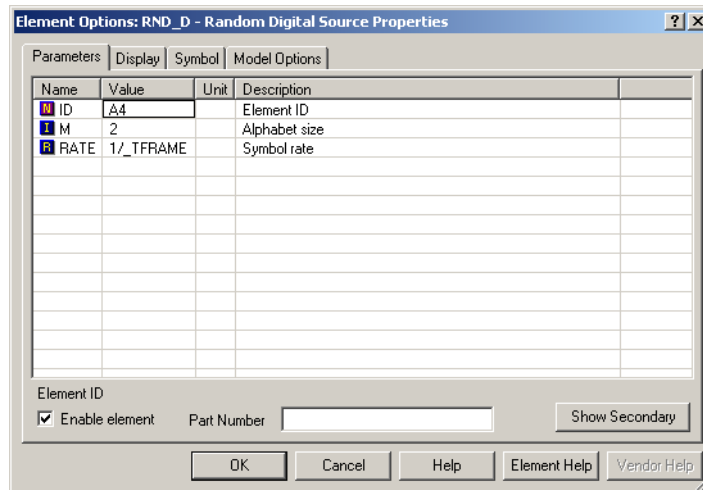


- 8 Click the **Add Test Point** button on the toolbar and add a Test Point (TP) between the QAM\_TX and AWGN blocks. Add another Test Point (TP) at the output of the RCVR block as shown in the following figure.
- 9 Connect the blocks and test points as shown in the following figure.

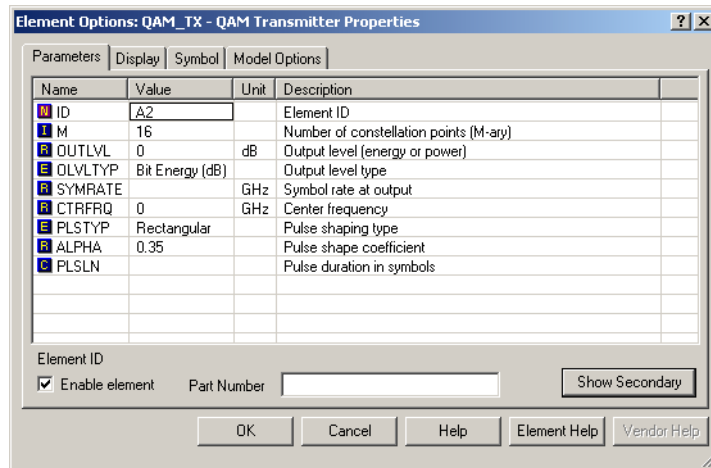


- 10 Double-click the RND\_D block in the system diagram and verify that the M parameter is **2**.

Because  $M = 2$ , RND\_D is set by default to generate a digital signal that varies between “0” and “1”. Leave all other secondary parameters at their default settings.

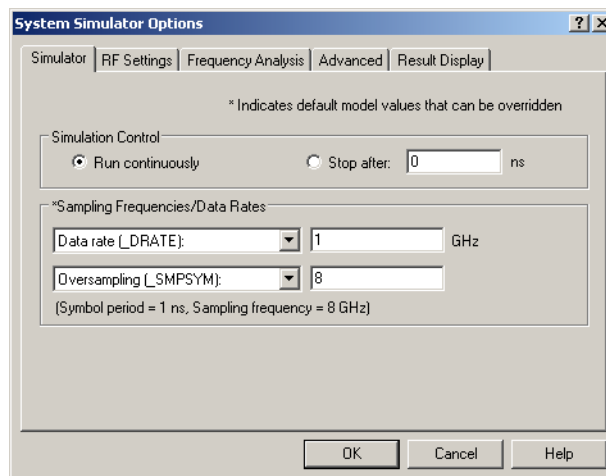


- 11 Double-click the QAM\_TX block and change the parameters as shown in the following figure.



In this dialog box you can control several parameters as well as the pulse shaping filter used on the in-phase and quadrature-phase signals.

- 12 RCVR automatically adjusts its parameters to agree with the transmitter parameters, so maintain the default settings.
- 13 Choose **Options > Default System Options**. The System Simulator Options dialog box displays. Verify that your settings match those in the following figure, then click **OK**.

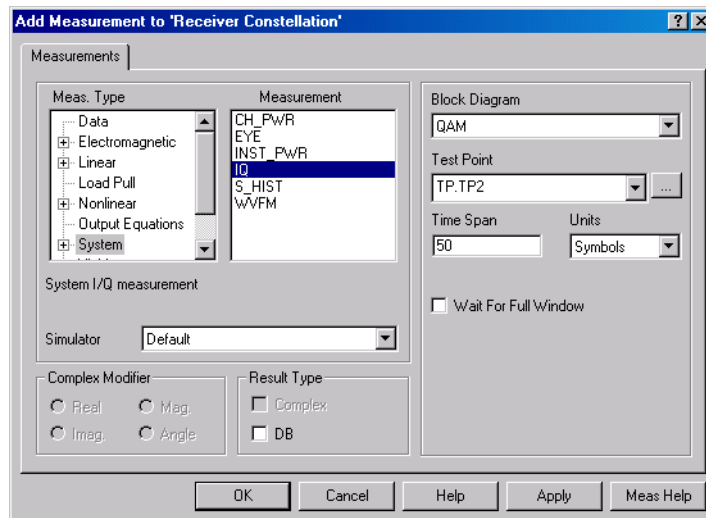


## Adding Graphs and Measurements

To add graphs and specify the measurements you want to plot:



- 1 In the Project Browser, right-click **Graphs** and choose **Add Graph**, or click the **New Graph** button on the toolbar. Type “**Complexbaseband**” in **Graph Name**. For **Graph Type**, select **Rectangular** and click **OK**.
- 2 Repeat step 1 to create a second graph named “**Receiver Constellation**”. For **Graph Type** select **Constellation** and click **OK**.
- 3 To view all of the windows, choose **Window > Tile Vertical**.
- 4 In the Project Browser, right-click “Complexbaseband” and choose **Add Measurement**. The Add Measurement dialog box displays.
- 5 Select **System** as the **Meas. Type** and select **WVFM** as the **Measurement**.
- 6 Select **TP. TP1** as the **Test Point**, and ensure that **Time Span** is “10” and **Units** is “**Symbols**”, then click **OK**.
- 7 In the Project Browser, right-click “Receiver Constellation” and choose **Add Measurement**.
- 8 Create an IQ measurement using the settings in the following figure, then click **OK**.

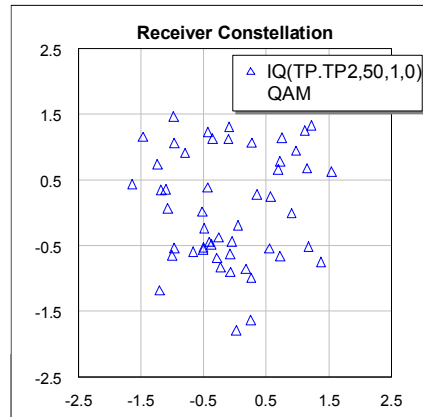
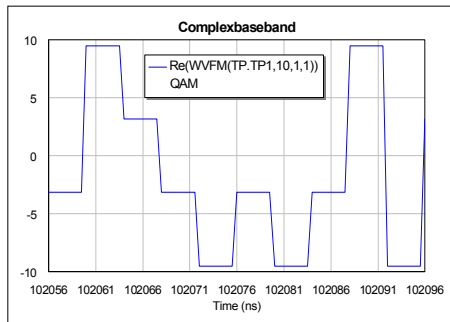


## Running the Simulation and Analyzing the Results

To run the simulation and configure the results display:



- 1 Choose **Simulate > Run/Stop System Simulators**. Let the simulation run for 3 seconds, then choose **Simulate > Run/Stop System Simulators** again to stop the simulation. The simulation responses in the following graphs should display.

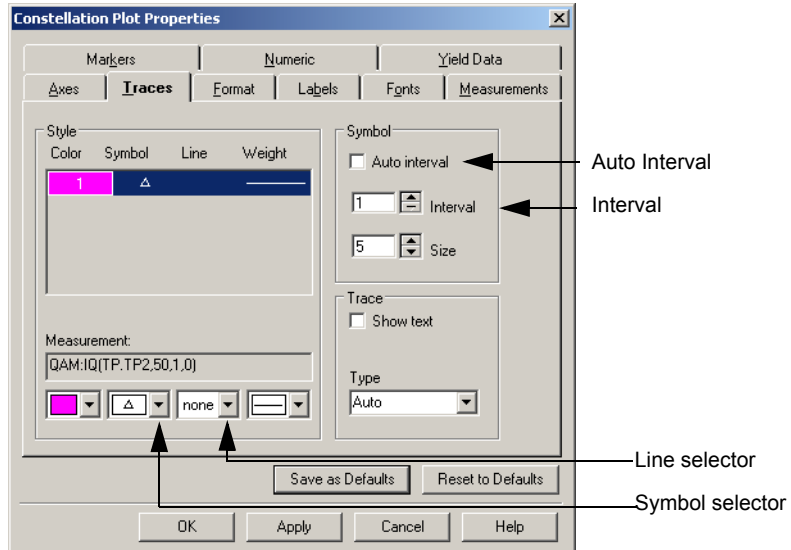


The received constellation does not appear as expected because the power spectral density of the noise source is set to 0 dB. Note that the time waveform of the complex baseband signal does not show eight samples per symbol as specified in the System Simulator Options dialog box.



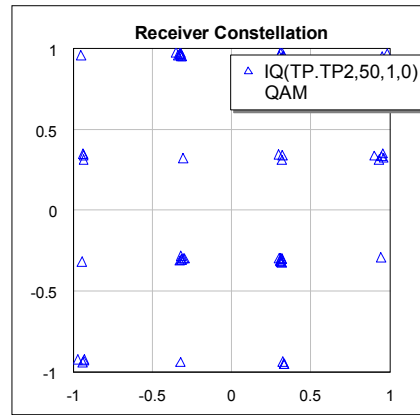
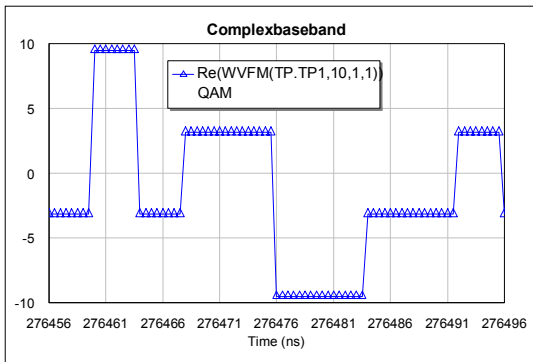
- 2 Select the Complexbaseband graph window and click the **Properties** button on the toolbar. The Rectangular Plot Properties dialog box displays.
- 3 Click the **Traces** tab.
- 4 Using the drop-down symbol and line selectors, specify a triangle as the symbol and a solid line as the line style, as shown in the following figure. (The lines will display on the Complexbaseband waveform).





- 5 Under **Symbol**, clear the **Auto interval** check box, change the **Interval** value to **1** and click **OK**.
- 6 On the system diagram, double-click the AWGN block and change the PWR parameter to “**-30**” dB, then click **OK**.
- 7 Start the simulation, let it run for about 10 seconds, and then stop the simulation.

Notice how the scatter plot changes. The simulation responses in the following graphs should display. On the Complexbaseband graph the triangular symbols display on the waveform. There are now eight samples per symbol. You can choose **Options > Default System Options** to return to the System Simulator Options dialog box and change the values under **Sampling Frequencies/Data Rates** to observe the different results.



Note that your graphs may not look identical to these examples due to printed size limitations.

**TIP:** Click the **Properties** button on the toolbar to edit the appearance of any graph, or right-click in the graph to zoom in and zoom out.

8 Choose **File > Save Project** to save your project.

## TUNING SYSTEM PARAMETERS

Visual System Simulator (VSS) architecture uses object-oriented techniques to enable fast and efficient simulations. A real-time tuner allows you to see the results of simulations as you tune parameters.

To tune a system parameter:

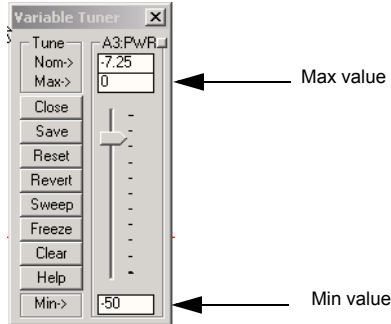


- 1 Click the system diagram window to make it active.
- 2 Click the **Tune Tool** button on the toolbar.
- 3 Position the cursor over the PWR parameter value of the AWGN block. The cursor displays as a small cross on a black background.
- 4 Click to activate the PWR parameter for tuning. The parameter value displays in blue. To disengage the Tune Tool, click anywhere in the design area.



- 5 To start the simulation, click the **Run/Stop System Simulators** button on the toolbar, then click the **Tune** button on the toolbar.

The Variable Tuner dialog box displays.



- 6 To observe the impact of the noise level, set the **Max** and **Min** values to “0” and “-50” respectively. Click the tuning bar and slide it to adjust the values while observing the results on the constellation graph.
- 7 Click the “x” at the upper right of the Variable Tuner dialog box to close it.
- 8 Stop the simulation by clicking the **Run/Stop System Simulators** button on the toolbar.
- 9 To de-tune the PWR parameter of the AWGN block, click the **Tune Tool** button on the toolbar and then click the PWR parameter value again. The parameter value displays in black. To disengage the Tune Tool, click anywhere in the design area.
- 10 Double-click the PWR parameter and change the value to 0.



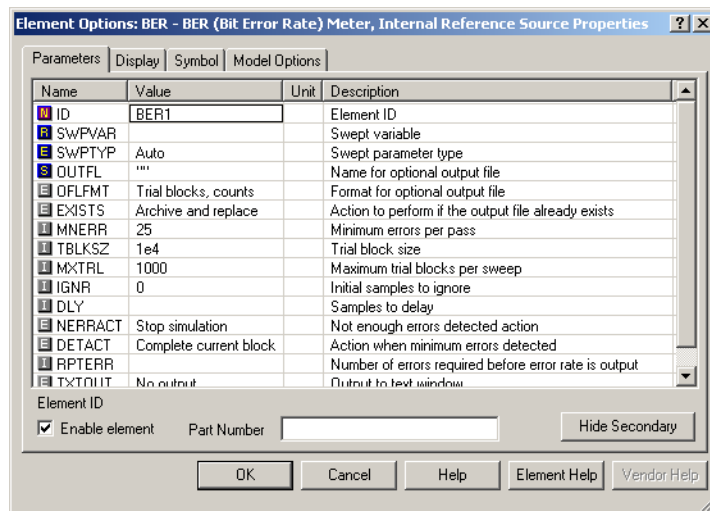
## CREATING A BER AND SER SIMULATION

System engineers often want to perform Bit Error Rate (BER) simulation. For some modulation schemes, it is easier to analytically calculate Symbol Error Rate (SER) than BER. This example shows how to generate both BER and SER. To generate a BER curve, a method of varying the signal-to-noise-ratio (SNR) must be available. For this purpose, a stepped variable “Eb\_No” is established to sweep out a range of values from 0 to 10, in increments of 1. The stepped variable is used as the value for the transmitter output power parameter, thus

raising the signal level as the variable is stepped. In this case, the AWGN power parameter is set to 0 db and the simulation is set up to increase the transmitter power.

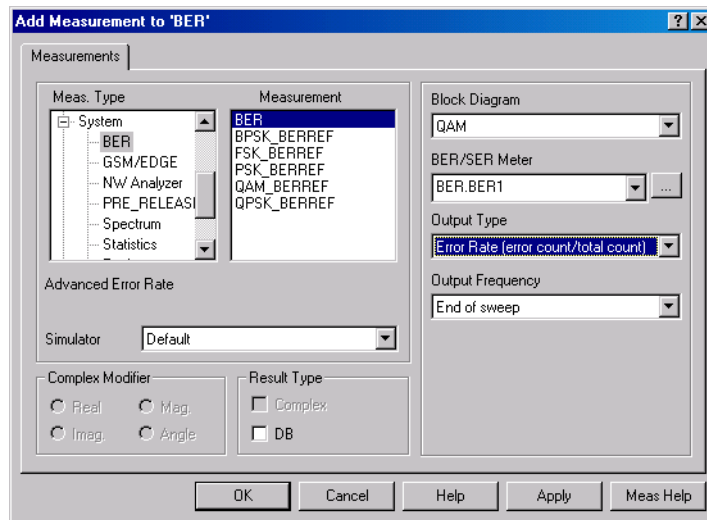
To create a BER simulation:

- 1 Click the system diagram window to make it active.
- 2 Choose **Draw > Add Equation** and move the cursor into the system diagram window. An edit box displays.
- 3 Position the edit box at the top of the system diagram and click to place it.
- 4 Type “**Eb\_NO = sweep (stepped(0,10, 1))**” (without the quotes) in the edit box, and then click outside of the box.
- 5 Double-click the QAM\_TX block and verify that the OUTLVL parameter is **Eb\_NO**, and the OLVL TYP parameter is **Bit Energy (dB)**, then click **OK**.
- 6 In the Element Browser under **System Blocks**, expand the **Meters** category and click the **BER** subgroup. Select the BER block (internal reference source) and place it to the right of the RCVR block in the system diagram.
- 7 Connect the BER block to the “D” node of the RCVR block.
- 8 Double-click the BER block. Click **Show Secondary** to view the secondary parameters. Edit the parameters to the values shown in the following figure, then click **OK**.

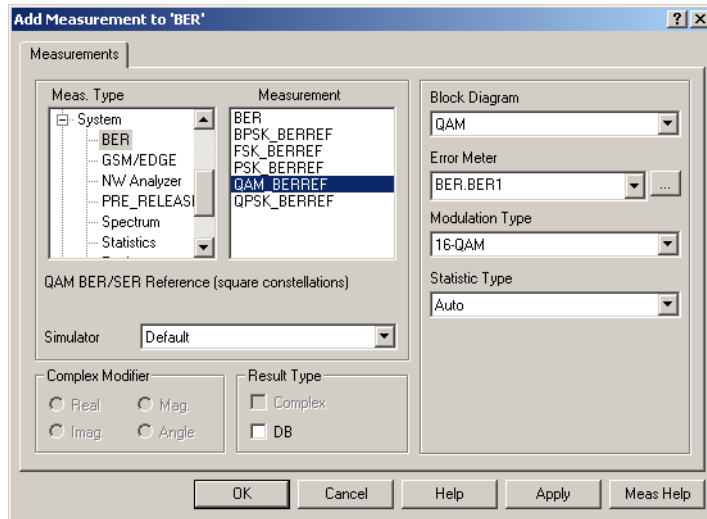


The BER block is now set to test  $1e7$  (MXTRL \* TBLKSZ) bits. It registers 25 errors before a BER computation is generated for each value of Eb\_NO. The BER block internally generates the original data source and compares the received bits to the transmitted bits. The last point on the BER curve (the 11th value of Eb\_NO) takes the longest to plot.

- 9 To add a BER plot to the project, add a rectangular graph named “**BER**”.
- 10 In the Project Browser, right-click “BER” and choose **Add Measurement**.
- 11 Create a BER measurement using the settings in the following figure, then click **Apply** to save the measurement.



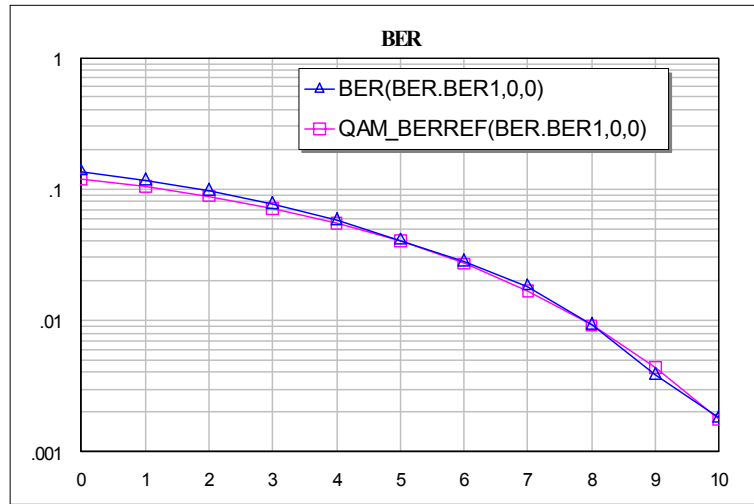
- 12 To verify the obtained results with the theoretical results, add another measurement to the BER graph using the settings in the following figure, then click **OK**.



- 13 Verify that the PWR parameter value of the AWGN block is **0dB**.
- 14 Select the BER graph window, then click the **Properties** button on the toolbar.
- 15 Click the **Axes** tab and set **Left1** to Log scale by selecting **Left1** under **Choose Axis** and selecting the **Log Scale** check box, then click **OK**.
- 16 Click the **Run/Stop System Simulators** button on the toolbar to start the simulation. As the simulation runs, the BER curve is generated. Note that the size of the received constellation becomes clearer as the power is increased or, as  $E_b/N_0$  is swept from 0 dB to 10 dB.

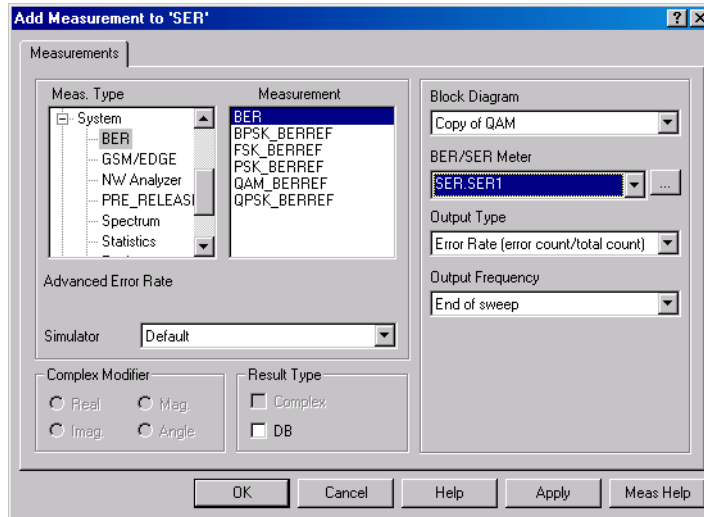


The simulation stops when 25 errors are counted at  $E_b/N_0 = 10\text{dB}$ . The simulation response in the following graph should display.

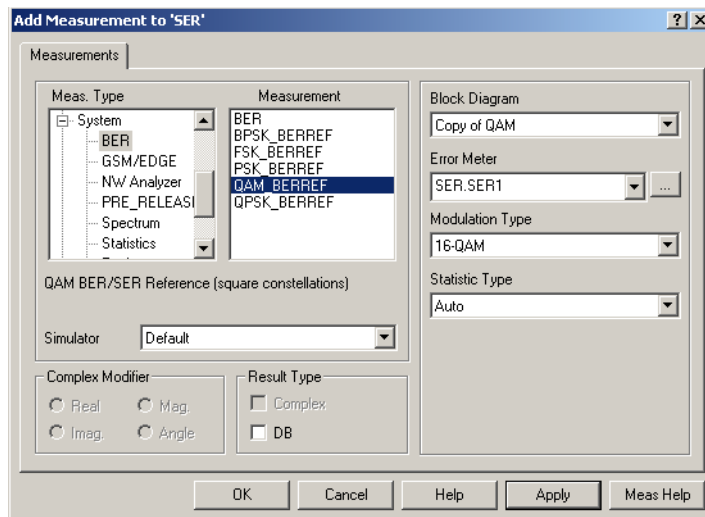


In this example the signal power was swept to plot the BER. You can sweep the noise power, keeping the signal power constant. Change the PWR parameter of the AWGN block to  $-E_b/N_0$  and the OUTLVL parameter of the QAM\_TX block to 0 to get the same BER curve achieved here.

- 17 Choose **File > Save Project**.
- 18 Similar to the BER version, you can create a SER vs  $E_b/N_0$  graph. In the Project Browser, click “QAM” under **System Diagrams**. Drag and drop the QAM icon onto the **System Diagrams** node. A “Copy of QAM” system diagram is created under **System Diagrams**.
- 19 Click the “Copy of QAM” window to make it active, then delete the BER block.
- 20 In the Element Browser, expand the **Meters** category, then click the **BER** subgroup. Select the SER block, drag it to the “Copy of QAM” system diagram, and connect it to the “D” node of the RCVR block.
- 21 Add a rectangular graph named “**SER**” to the project.
- 22 Add a measurement to the “SER” graph using the settings in the following figure.



- 23 Add another measurement to the “SER” graph using the settings in the following figure.

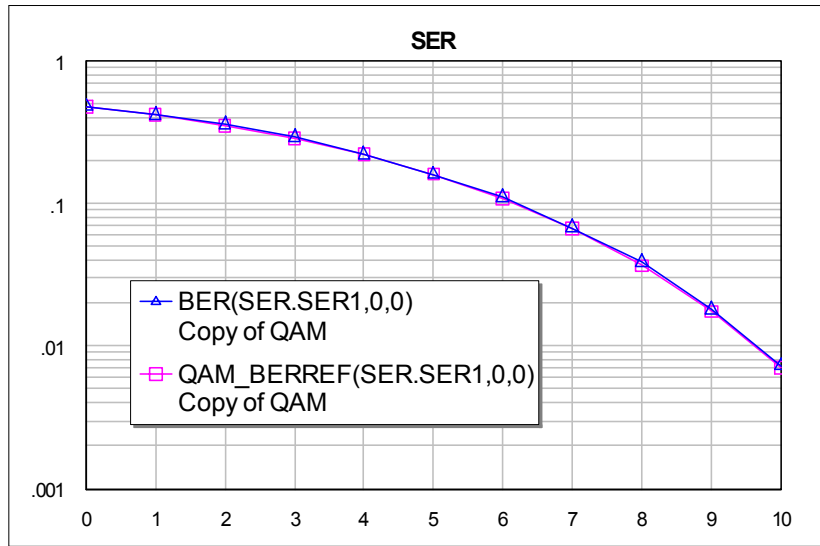


- 24 Select the “SER” graph window, then click the **Properties** button on the toolbar.
- 25 Click the **Axes** tab and set **Left1** to Log scale by selecting **Left1** under **Choose Axis** and selecting the **Log Scale** check box, then click **OK**.





- 26 Click the **Run/Stop System Simulators** button on the toolbar to start the simulation. As the simulation runs, the SER curve is generated. The simulation response in the following graph should display.

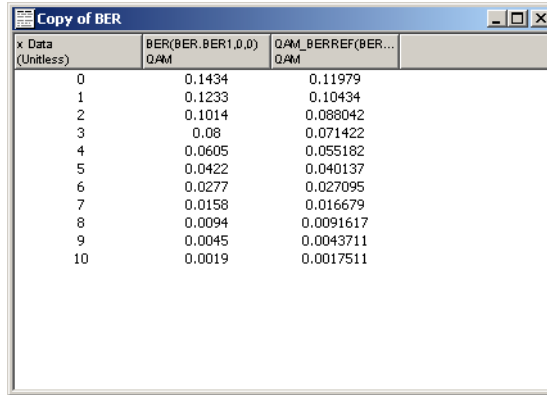


You can also plot BER and SER against  $E_s/N_0$  by specifying  $E_s/N_0$  as the SWPTYP parameter in BER and SER blocks.

## Converting BER Curve Results to a Table

To convert the results of the BER curve to a table:

- 1 In the Project Browser select the “BER” graph, right-click and choose **Duplicate as > Table**. A new table (graph) named “Copy of BER” displays under **Graphs**.



The screenshot shows a window titled "Copy of BER" with a table of data. The table has three columns: "x Data (Unitless)", "BER(BER.BER1,0,0) QAM", and "QAM\_BERREF(BER... QAM". The data is as follows:

x Data (Unitless)	BER(BER.BER1,0,0) QAM	QAM_BERREF(BER... QAM
0	0.1434	0.11979
1	0.1233	0.10434
2	0.1014	0.088042
3	0.08	0.071422
4	0.0605	0.055182
5	0.0422	0.040137
6	0.0277	0.027095
7	0.0158	0.016679
8	0.0094	0.0091617
9	0.0045	0.0043711
10	0.0019	0.0017511

- 2 To change the numerical precision of the table, select the “Copy of BER”, graph window and click the **Properties** button on the toolbar. Make any desired changes in the Tabular Graph Format dialog box, then click **OK**.
- 3 Choose **File > Save Project**.

## ADDING AN MWO SUBCIRCUIT TO A SYSTEM

---

The circuit simulation capabilities of Microwave Office (MWO) and Visual System Simulator (VSS) provide a unique environment in which to measure the impact of RF components on system performance. Measurements you can make include, for example, the impact of phase noise on BER, spectral regrowth due to the non-linearities of an amplifier, and the impact of filter characteristics on BER.

This exercise presents features of the VSS environment and demonstrates its integration with the MWO circuit simulation environment. You add an actual MWO filter circuit to the QAM system that you built in the previous example, and then you measure the impact of the filter on BER performance as you change filter parameters.

The procedures in this example include:

- Importing an MWO ideal filter circuit to the system project
- Approximating the filter response before including it in the simulation
- Changing filter parameters and monitoring the impact on BER performance
- Creating a power spectral density plot.

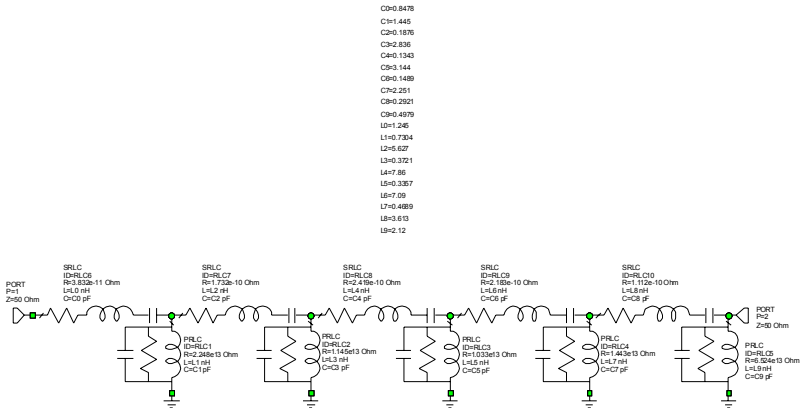
## ADDING AN MWO FILTER CIRCUIT TO THE SYSTEM

---

In this example you import an MWO ideal filter schematic into your existing QAM project. Although there are also ideal filter blocks available in VSS, the purpose of this exercise is to demonstrate how you can implement an MWO structure in VSS. The complete example is available as “QAM+Filter.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory. You can use this example file as a reference.

- 1 Open your QAM project if it is not already open. (Choose **File > Open Project** and select the directory in which you saved “QAM.emp”). If you did not build this project, you can get the project file from the *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory.

- In the Project Browser, right-click **Circuit Schematics** and choose **Import Schematic**. Select the “Ideal filter.sch” schematic from the *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory. The following filter schematic displays.

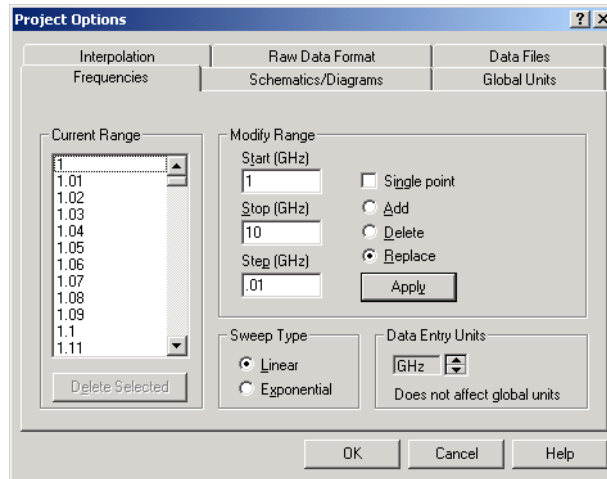


## Testing the Filter

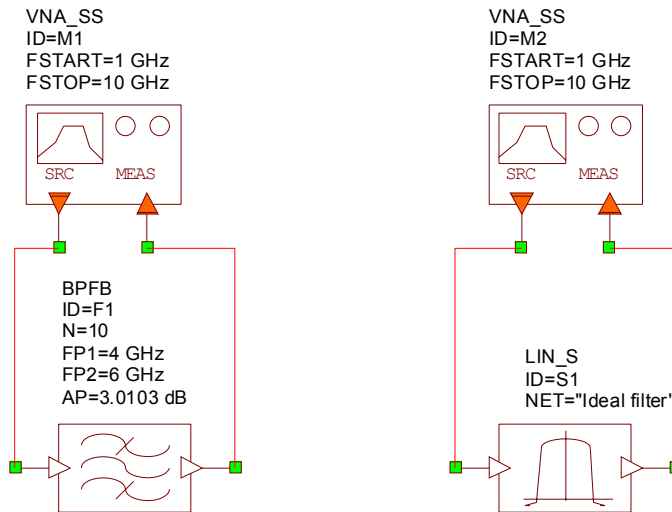
You should verify the filter response before including the filter in the system simulation. You will use a separate system diagram to connect the MWO filter to a VNA\_SS block and plot the frequency magnitude and phase response on a graph to check the frequency response of the filter before including it in a simulation chain.

To verify the filter response:

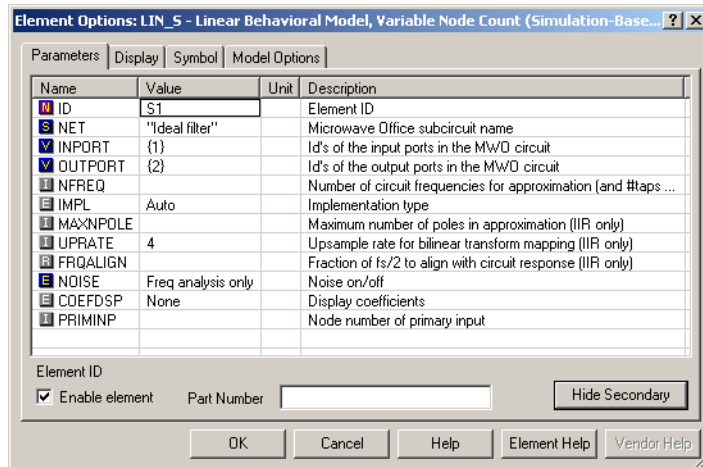
- Choose **Options > Project Options** and click the **Frequencies** tab to set the project frequency.
- Specify the **Start**, **Stop** and **Step** values shown in the following figure, click **Apply** to display the values in **Current Range**, and then click **OK**.



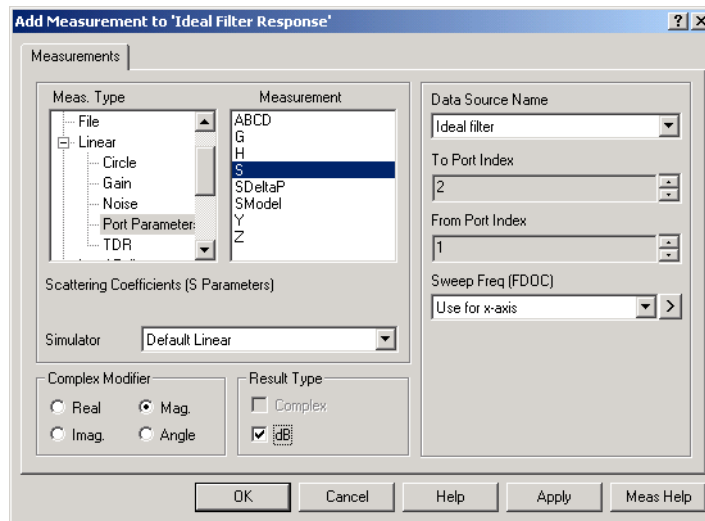
- 3 Right-click **System Diagrams** and choose **New System Diagram** to add a diagram named “Filter Test”.
- 4 Click the “Filter Test” window in the workspace to make it active.
- 5 In the Element Browser, expand the **Filters** category, then click the **Bandpass** subgroup. Select the BPFb block and place it on the system diagram as shown in the following figure.
- 6 Expand the **RF Blocks** category, then the **Linear Filters** category, then click the **Simulation Based** subgroup. Select the LIN\_S block and place it on the system diagram as shown in the following figure.
- 7 Expand the **Meters** category, then click the **Network Analyzers** subgroup. Select the VNA\_SS block, place it on the system diagram, and connect it across the BPFb block as shown in the following figure.
- 8 Repeat step 7 to connect a VNA\_SS block across the LIN\_S block as shown in the following figure.



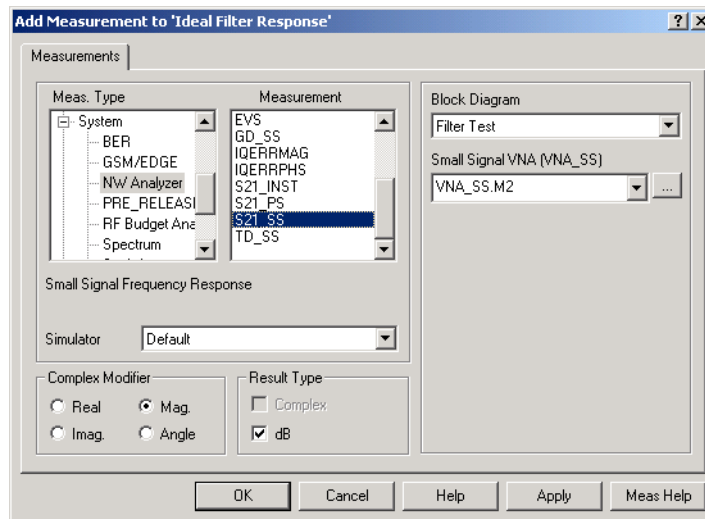
- 9 In the BPFB block, set the N parameter to “10”, the FP1 parameter to “4” GHz, and the FP2 parameter to “6” GHz.
- 10 In both of the VNA\_SS blocks set the FSTART parameter to “1” and the FSTOP parameter to “10” GHz.
- 11 In the LIN\_S block set the parameters as shown in the following figure.



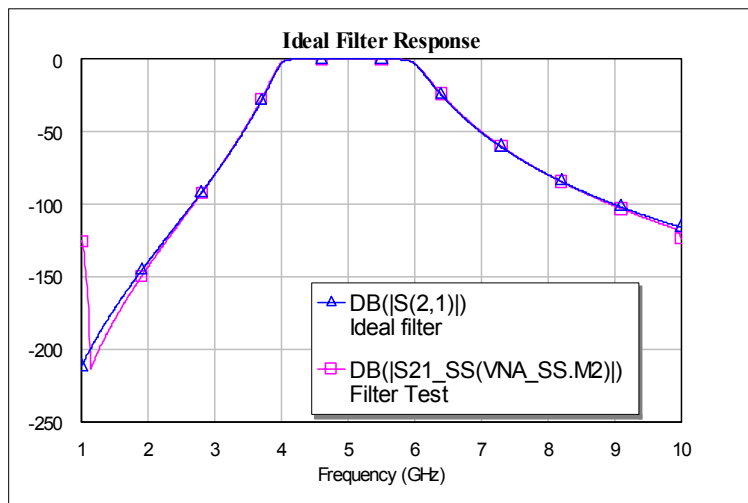
- 12 Create a rectangular graph named “**Ideal Filter Response**” and add a measurement to the graph using the settings in the following figure, then click **Apply**.



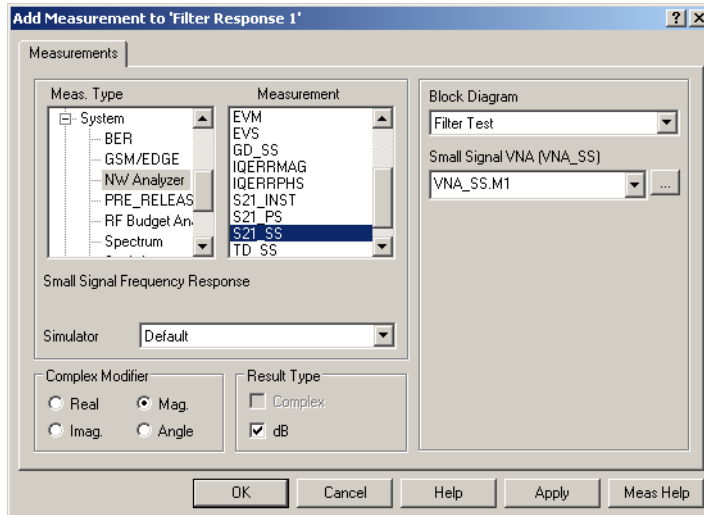
- 13 Add another measurement to the same graph using the settings in the following figure, then click **OK**.



- 14 Run both the Harmonic Balance simulator and System simulator. The simulation response in the following graph should display.



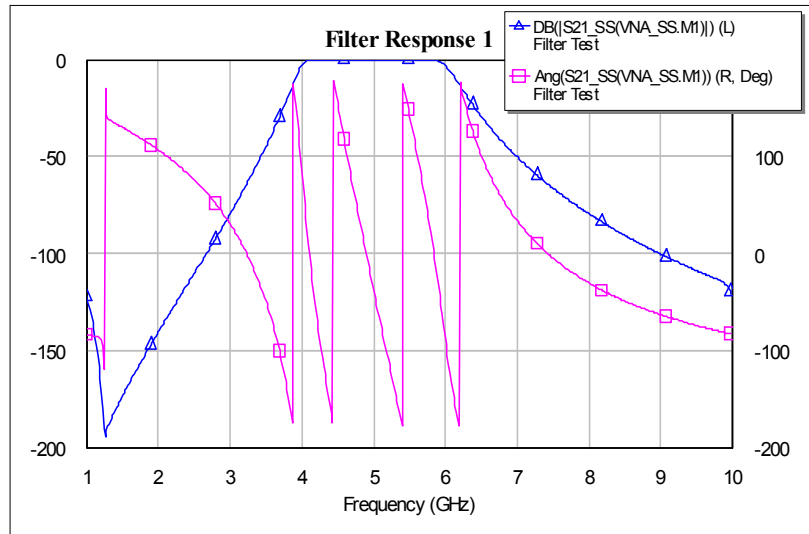
- 15 Create another rectangular graph named “**Filter Response 1**” and add a measurement to the graph using the settings in the following figure, then click **Apply**.



- 16 Add another measurement to the same graph except select **Angle** as the **Complex Modifier** and deselect **DB** as Result type, then click **OK**.
- 17 Run the System Simulator. The simulation response in the following graph should display after making some axes changes: Double-click the legend in



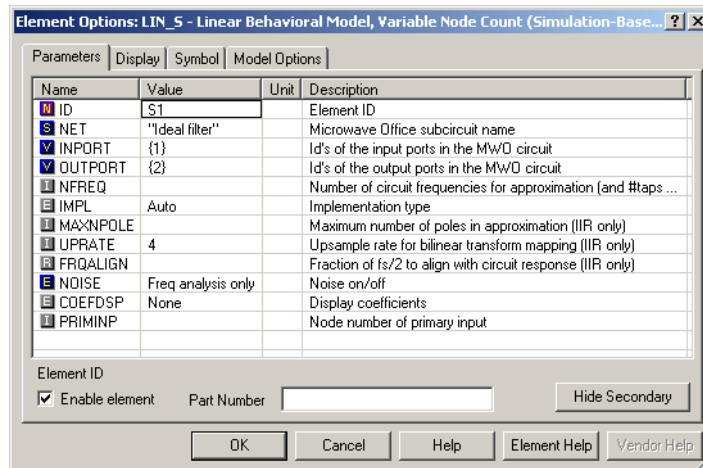
the graph. In the Rectangular Plot Properties dialog box, click the **Measurements** tab. Under **Select measurement to edit** select **Filter Test: (Ang(S21\_SS(VNA\_SS.M1)))** and under **Choose axis** select **Right 1**. Click **Apply**, then **OK**.



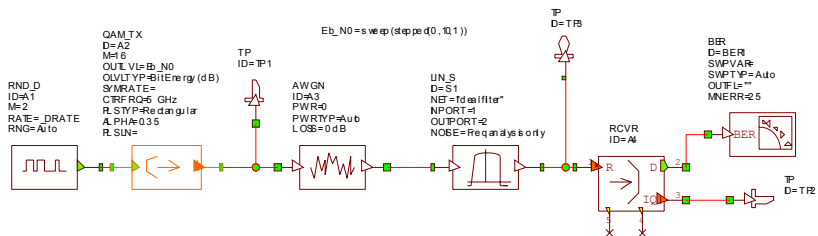
## Simulating the QAM System

To simulate the QAM system:

- 1 On the “QAM” system diagram, double-click the QAM\_TX block CTRFRQ parameter and set the value to “5” GHz.
- 2 In the Element Browser, expand the **RF Blocks** category, then expand the **Linear Filters** category and click the **Simulation Based** subgroup. Select the LIN\_S block and place it on the system diagram between the AWGN and RCVR blocks. Connect the blocks, moving them as necessary.
- 3 Change the LIN\_S block NET parameter to “**Ideal filter**” (including quotes) as shown in the following figure, then click **OK**.



- 4 Add a test point at the output of the LIN\_S block as shown in the following figure.

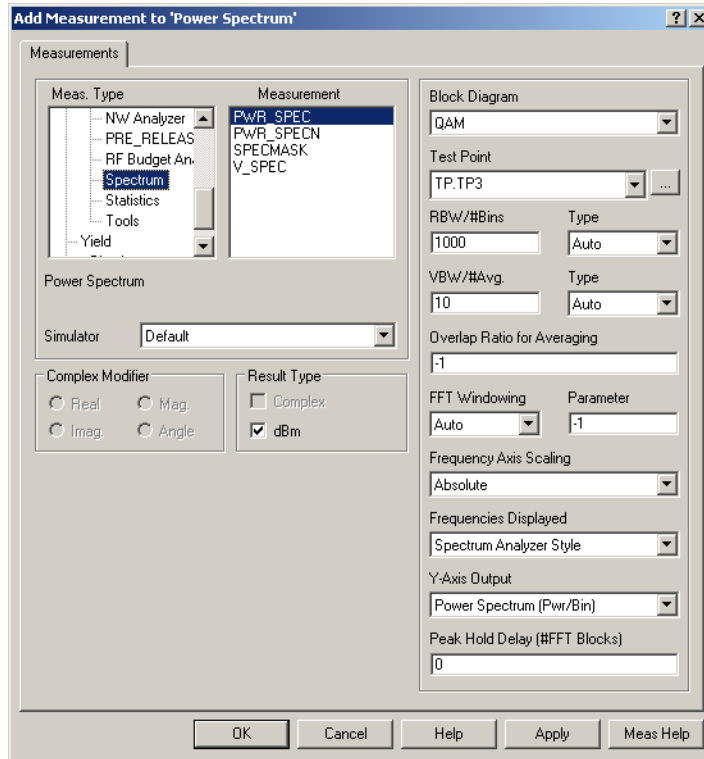


## Adding a Graph and Analyzing the Results

You now add a graph and two measurements to create an overlay to view the power spectral density for the QAM system before and after the addition of the bandpass filter.

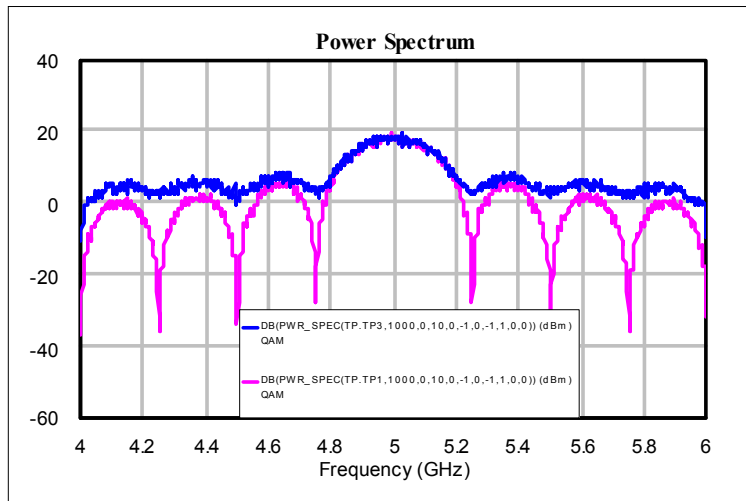
To add a graph and measurements:

- 1 Add a rectangular graph named **"Power Spectrum"**.
- 2 Add a PWR\_SPEC measurement using the settings in the following figure, then click **Apply**.



Specifying “**TP.TP3**” for **Test Point** places the measurement after the LIN\_S block.

- 3 Add another PWR\_SPEC measurement with the same settings, but choose **TP.TP1** (the test point prior to the AWGN block) for **Test Point**, then click **OK**.
- 4 Run the simulation. Note that the BER performance has slightly degraded. The simulation response in the following graph should display.



- 5 Save the project as “**QAM+Filter**”.

## Experimenting with Filters

Try varying the filter parameters using the Variable Tuner to observe the impact on the system BER performance. Remember to restore the filter parameters to their current values for the next exercise, however.

Note that as you change the filter parameters, you can improve or severely degrade the system’s BER performance. You may also want to experiment with other filters. In the next example you will discover that the primary reason for the degradation in BER performance is the impact of the filter’s phase response.

# USING AN MWO NONLINEAR ELEMENT IN VSS

This chapter demonstrates how to use the Microwave Office (MWO) Harmonic Balance nonlinear simulator with a Visual System Simulator (VSS) simulation. In this example you simulate an amplifier model and then measure the impact of the amplifier on the overall system. You observe the resulting power spectrum, the constellation graph, eye diagram, and the group delay. You also overlay a gain plot of the filter response with power spectral density of the transmitted signal, and analyze the simulation results.

The procedures in this example include:

- Importing an MWO amplifier model into a system project
- Working with the VSS Vector Signal Analyzer block
- Creating plots for eye diagrams, filter response, and group delay

## IMPORTING AN AMPLIFIER MODEL INTO VSS

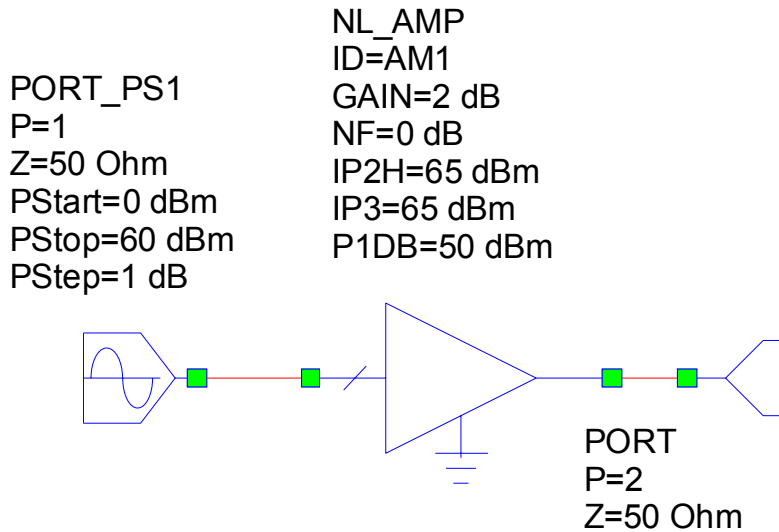
---

In this example you add an amplifier model to the QAM system built in the previous chapter. The complete example is available as “QAM+Filter+Amp.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory. You can use this example file as a reference.

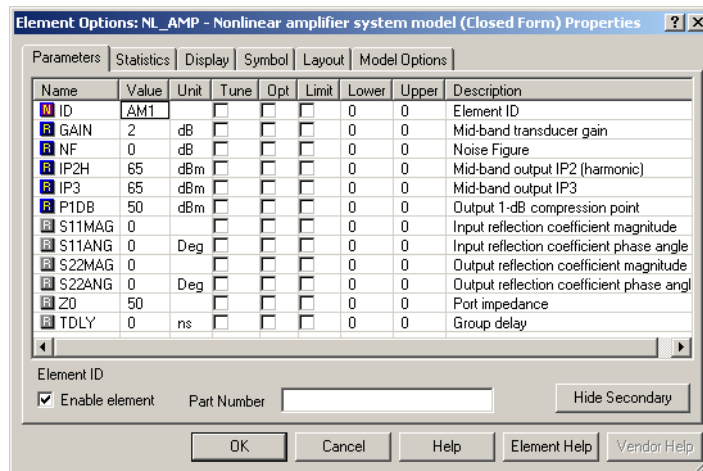
To import an amplifier:

- 1 Open the “QAM+ Filter.emp” project you saved in the previous chapter if not already open. If you did not build this project, you can get the project file from the *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory.
- 2 In the Project Browser, right-click **Circuit Schematics** and choose **Import Schematic**. Select the “amplifier.sch” schematic from the *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\QAM* directory.

The following amplifier schematic displays in the project.

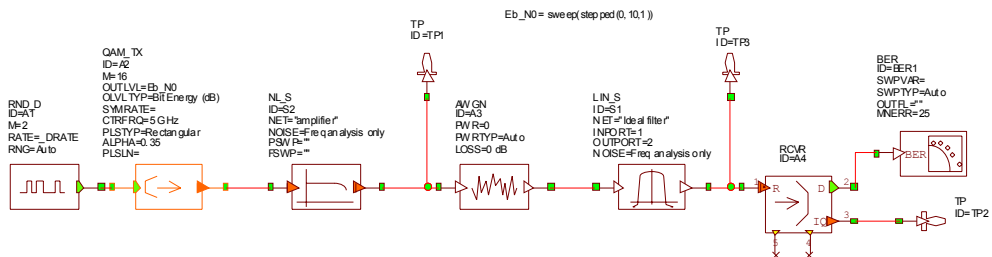
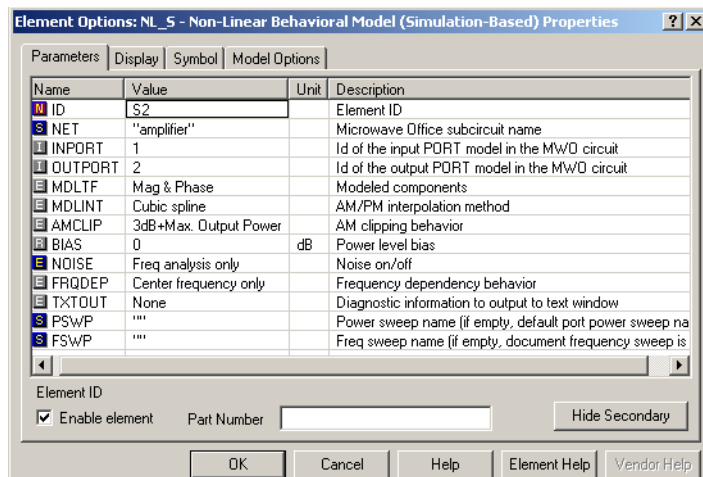


- Verify that the NL\_AMP parameters are set to the following values, then click **OK**.



- In the QAM system diagram, separate the QAM\_TX block and the TP1 test point and disconnect the wire between them.

- 5 In the Element Browser under **System Blocks**, expand the **RF Blocks** category, then expand the **Amplifiers** category and click the **Simulation Based** subgroup. Select the NL\_S block and place it between the QAM\_TX and AWGN blocks as shown in the following figure.
- 6 Connect the NL\_S block to the QAM\_TX and AWGN blocks.
- 7 Double-click the NL\_S block and set its NET parameter to “**amplifier**” to reference the amplifier schematic (include the quotes), then click **OK**.



For information about propagated parameters such as SNR, as well as group delay and phase rotation, see the online Help for the System Tools measurements in the VSS Measurements Reference.

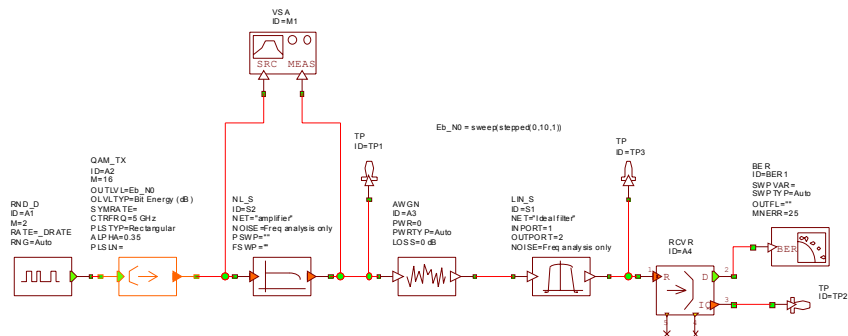
- 9 Save the project as “QAM+Filter+Amp”.

## Using the Vector Signal Analyzer Block

Next you use the Vector Signal Analyzer block to plot the instantaneous output power of the QAM transmitter relative to the AM/AM characteristics of the amplifier.

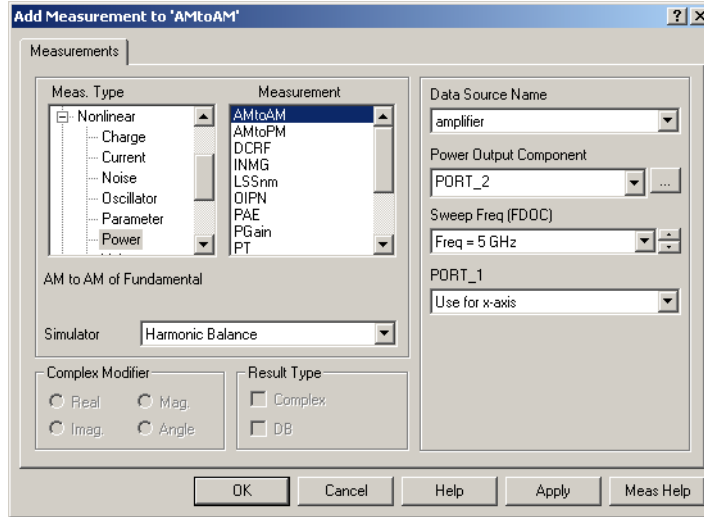
To use the VSA block to plot output power:

- 1 In the **System Blocks** list expand the **Meters** category, then click the **Network Analyzers** subgroup. Select the VSA block and place it above the NL\_S block in the QAM system diagram.
- 2 Connect the two ports of the VSA block to either side of the NL\_S block as shown in the following figure.

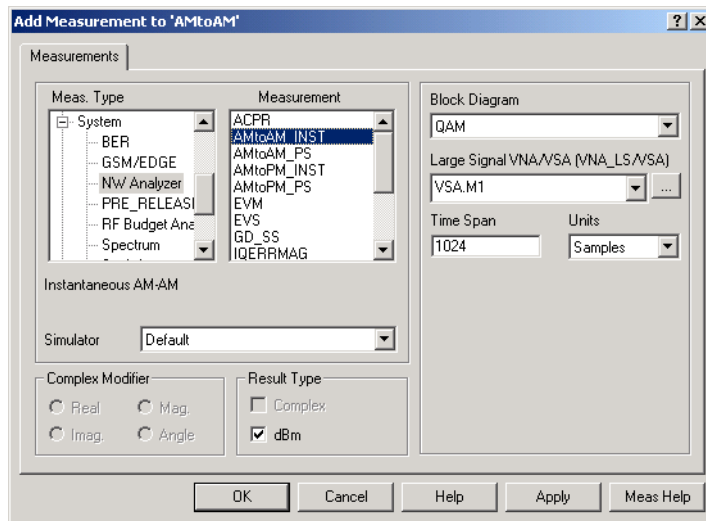


- 3 Save the project.
- 4 Add a rectangular graph named “AMtoAM.”
- 5 Add a measurement to the graph using the following settings, then click **Apply**.

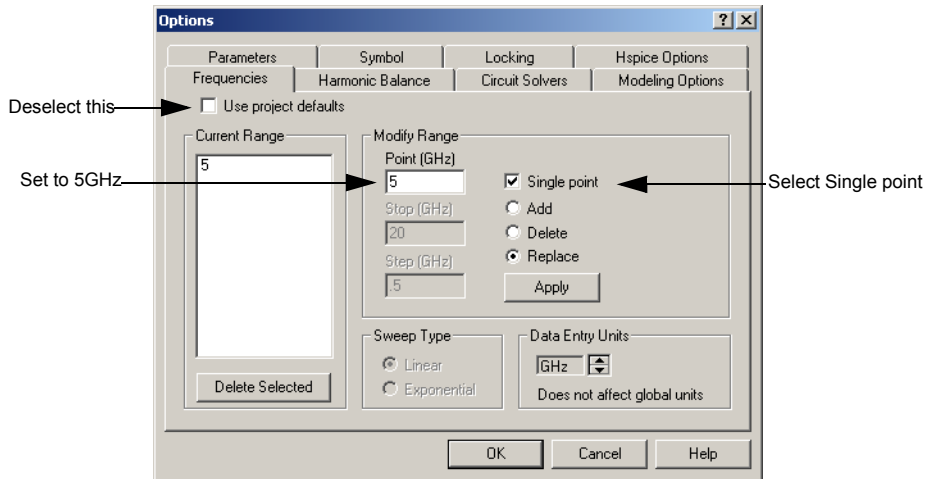




- 6 Add another measurement to the graph using the following settings, then click **OK**.



- 7 In the Project Browser, right-click the “amplifier” schematic and choose **Options** to display the Options dialog box.
- 8 Click the **Frequencies** tab and set the options as shown in the following figure, then click **OK**.



- 9 Start the harmonic balance simulation by clicking the **Analyze** button on the toolbar. This updates the AMtoAM graph with harmonic balance results.

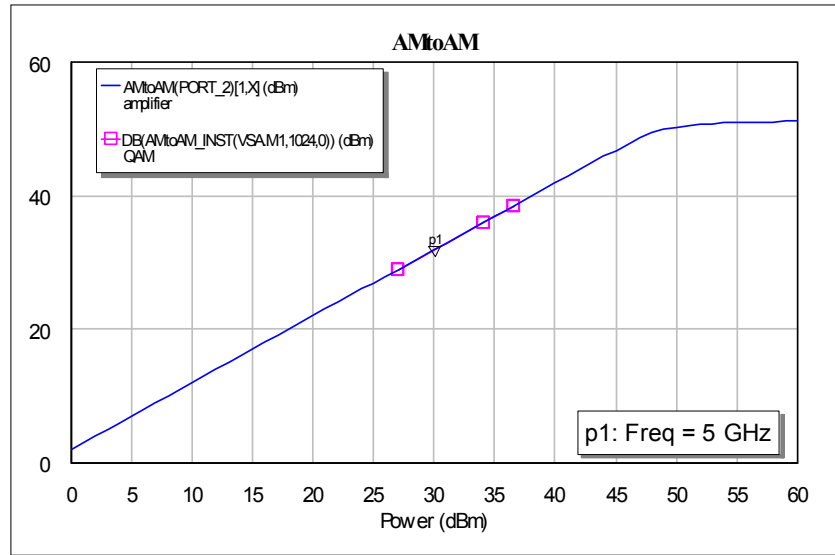


- 10 For best graph appearance, select the AMtoAM graph window, click the **Properties** button on the toolbar, click the **Traces** tab, and change the symbol from a triangle to **none**.



- 11 Start the system simulation by clicking the **Run/Stop System Simulators** button on the toolbar. As the simulation runs, a marker moves along the plot that resulted from the harmonic balance simulation. The marker moves because you are sweeping the Eb\_NO variable. The marker indicates the operating point of the system relative to the 1dB compression point of the amplifier. As you decrease the value of the 1dB compression point, the operating point of this system moves into the nonlinear region of the amplifier.

The simulation response in the following graph should display.



For information on the LIN\_S, NL\_S and VSA blocks, see the online Help for the **Meters** and **RF Blocks** categories of System Blocks in the VSS System Block Catalog.

Note that these examples use ideal behavioral models for the filter and amplifier. You can also use actual circuit models for the filter and amplifier.



# RF BUDGET ANALYSIS

This chapter demonstrates how to perform RF Budget Analysis in Visual System Simulator (VSS). In this example you simulate an RF chain and analyze a Cascaded Noise Figure over Cascaded Operating gain.

The procedures in this example include:

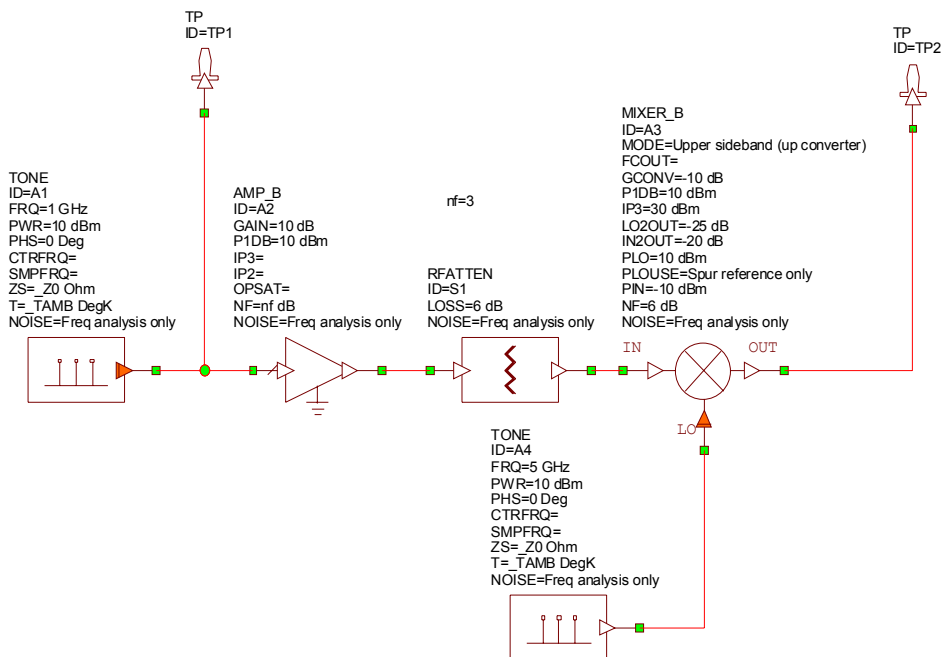
- Creating an RF chain
- Setting up measurements
- Performing yield analysis

## Creating an RF Chain

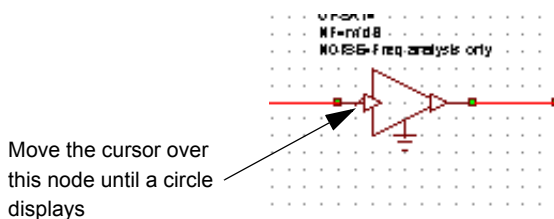
The complete example is available as “RF Budget Analysis” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\RF Budget* directory. You can use this example file as a reference.

To create an RF chain:

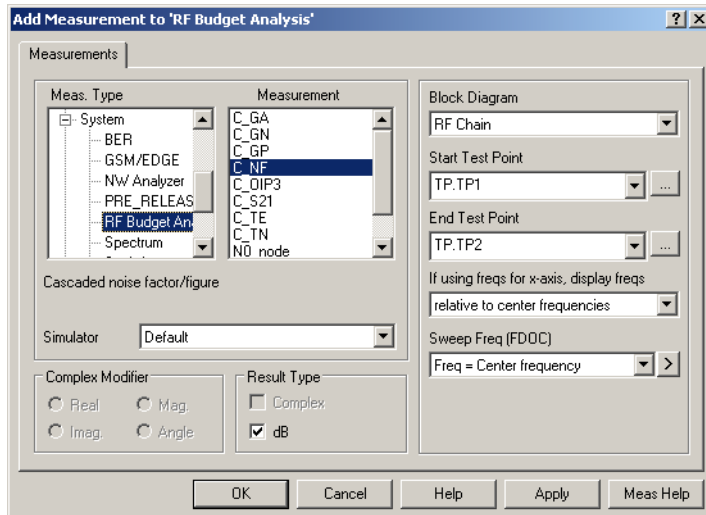
- 1 Create a new project named “**RF Budget Analysis**”.
- 2 Create a system diagram named “**RF Chain**”.
- 3 Complete the system diagram as shown in the following figure. All of the system blocks are located in subgroups of the **RF Blocks** category. TONE is under **Sources**, AMP\_B is under **Amplifiers**, RFATTEN is under **Other**, and MIXER\_B is under **Mixers**.



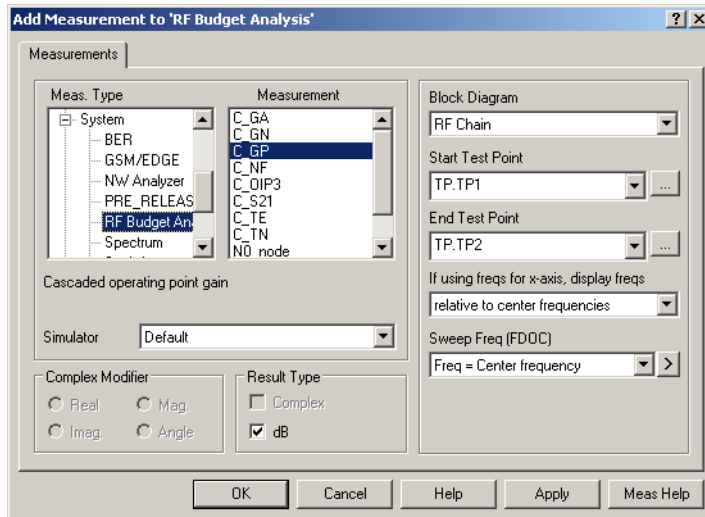
- 4 Add an equation to the system diagram as “**nf=3**”.
- 5 Assign “**nf**” to the NF parameter of the AMP\_B block.
- 6 To change the data type for AMP\_B, move the cursor over the block as shown in the following figure. Double-click when a circle displays around the node. The System Node Settings window displays.



- 7 Select **Complex** as the **Node data type** and click **OK**.
- 8 Add a rectangular graph named “**RF Budget Analysis**”.
- 9 Add a cascaded noise figure measurement as shown in the following figure.



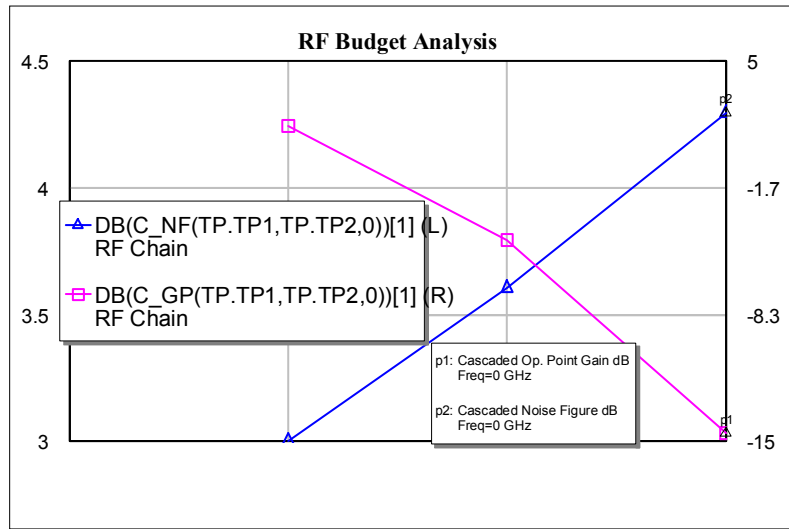
- 10 Add a cascaded operating point gain measurement as shown in the following figure.



- 11 In the “RF Budget Analysis” graph properties, set the C\_GP measurement to display on the Right axis.
- 12 Click the **Analyze** button on the toolbar.



The simulation response shown in the following graph should display.



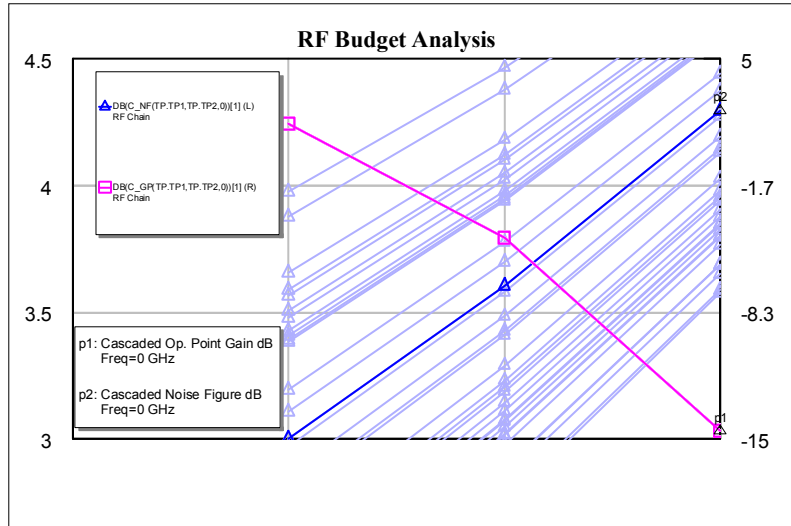
- 13 Click the **Variables** button on the toolbar. In the Variables dialog box, for the nf parameter select the **Use Statistics** check box, set **Tolerance** to "4" and **Distribution** to **Uniform** as shown in the following figure. If these columns do not display, right click any column heading and choose from the list that displays. Close the Variables dialog box.

The screenshot shows the "Variables" dialog box with the "nf" parameter selected. The "Use Statistics" checkbox is checked, and the "Tolerance" is set to 4. The "Distribution" is set to Uniform. The table below lists the variables and their configurations.

Schematic	Element	ID	Parameter	Value	Use Statistics	Tolerance	Distribution
RF Chain		@R...	nf	3	<input checked="" type="checkbox"/>	4	Uniform
RF Chain	TONE	A1	NOISE	0	<input type="checkbox"/>	0	Normal
RF Chain	TONE	A1	PHS	0	<input type="checkbox"/>	0	Normal
RF Chain	TONE	A1	PWR	10	<input type="checkbox"/>	30	Normal
RF Chain	TONE	A1	FRQ	1	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	OVFLTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	MSKTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	FRQAX	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	WNDWHN	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	SLDFRC	0.5	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP2	WNDTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	OVFLTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	MSKTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	FRQAX	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	WNDWHN	0	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	SLDFRC	0.5	<input type="checkbox"/>	0	Normal
RF Chain	TP	TP1	WNDTYP	0	<input type="checkbox"/>	0	Normal
RF Chain	TONE	A4	NOISE	0	<input type="checkbox"/>	0	Normal
RF Chain	TONE	A4	PHS	0	<input type="checkbox"/>	0	Normal
RF Chain	TONE	A4	PWR	10	<input type="checkbox"/>	30	Normal
RF Chain	TONE	A4	FRQ	1	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	TXTOU	0	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	MXUPSMP	1000	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	SPURBW	100	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	SIGBW	100	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	INHMAX	5	<input type="checkbox"/>	0	Normal
RF Chain	MIXER_B	A3	INHMAX	5	<input type="checkbox"/>	0	Normal



- 14 Choose **Simulate > Yield Analysis**. The Yield Analysis dialog box displays. Ensure that the **Analysis Method** is **Yield Analysis**.
- 15 Click **Start** to start yield analysis. The response in the following graph should display. To stop the analysis at any time click **Stop**.



- 16 To change the display of the yield data, open the graph properties window and click the **Yield Data** tab to specify settings.



## VSS EXAMPLES

---

This chapter includes additional useful examples such as FSK, Phase Noise and I/Q Imbalance, End-to-End QAM system, and mixer modeling. It also includes steps for displaying measurements such as ACPR, EVM, and others.

### FSK EXAMPLE

---

In this example you build and simulate a complete transmitter-channel-receiver chain for a Binary Frequency Shift Keying (BFSK) transmission. The example shows how to generate a frequency shift keying (FSK) source using elementary blocks or a “black box” FSK modulator (called FSK\_SRC) in VSS. It is not always necessary to construct a receiver and transmitter using elementary blocks. For many common modulation methods, corresponding black boxes already exist in VSS. This exercise, however, is intended to show that using the black box or creating one using the elementary blocks produces identical results. You will also construct an FSK demodulator using basic blocks, and verify the performance of the system by setting different parameters. For more information and application notes on FSK modulation please visit our website at <http://www.appnwave.com/>.

The procedures in this example include:

- Verifying the BFSK waveform generated using different methods
- Channel and noise scaling and system performance
- Monitoring the BER and sweep statistics in a text window.

### Using a “Black Box” FSK Modulator

In this section you create a project and BFSK system diagram, then use the FSK modulator (FSK\_SRC) and plot the power spectrum of the modulated signal. The complete example is available as “bfsk.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\BFSK* directory.

To generate an FSK source using a “black box” FSK modulator:

- 1 Create a project named “**bfsk**”.

- 2 Add a system diagram named “**BFSK**” to the project.
- 3 In the Element Browser **System Blocks** list, expand the **Modulation** category, then click the **FSK** subgroup. Select the FSK\_SRC block and place it on the system diagram. Verify that its parameters are set to the following values, then click **OK**.

Element Options: FSK\_SRC - FSK Modulated Signal Properties

Parameters | Display | Symbol | Model Options

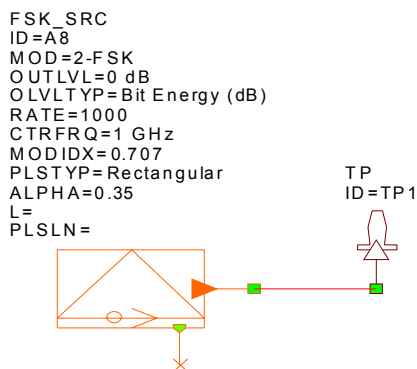
Name	Value	Unit	Description
ID	A8		Element ID
MOD	2-FSK		FSK modulation
OUTLVL	0	dB	Output level (energy or power)
OLVLTYP	Bit Energy (dB)		Output level type
RATE	1000		Symbol rate
CTRFRQ	1	GHz	Center frequency
MODIDX	0.707		Modulation index
PLSTYP	Rectangular		Frequency shape pulse type
ALPHA	0.35		Pulse shape coefficient
L			Pulse length 'L' in symbols
PLSLN			Pulse duration in symbols
IPHS	0	Deg	Initial phase
ISYM			Initial symbol state
STRTTYP	At Symbol 0		Startup type
DTYP	M-Ary		Digital source type
_SMPSYM	_SMPSYM		Samples per symbol
RSEED	{0}		Digital source random number generator seeds
_BLKSZ	_BLKSZ		Symbols per pass

Element ID

☒ Enable element Part Number  Hide Secondary

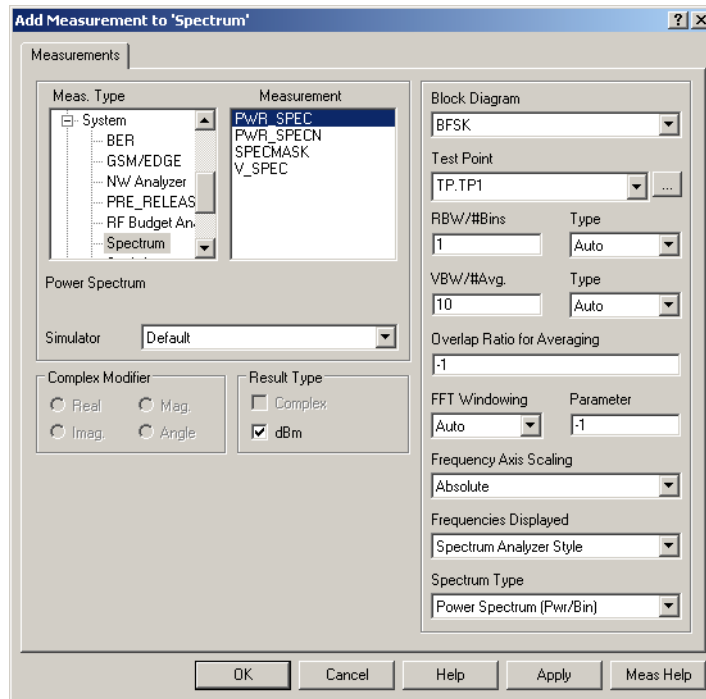
OK Cancel Help Element Help

- 4 Add and connect a test point as shown in the following figure.

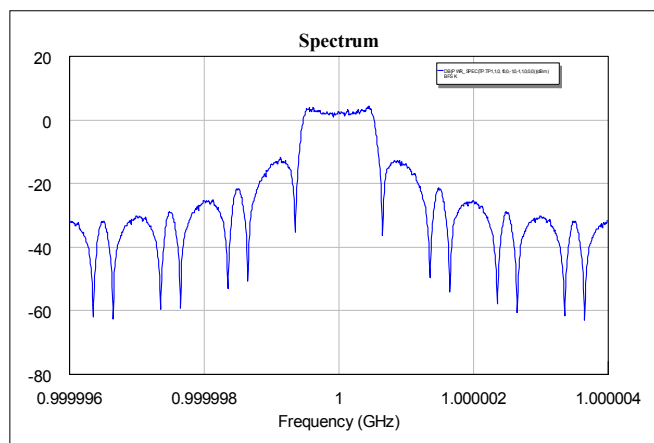


- 5 Add a rectangular graph named “**Spectrum**”.

- 6 Add a measurement to the graph using the settings in the following figure, then click **OK**.



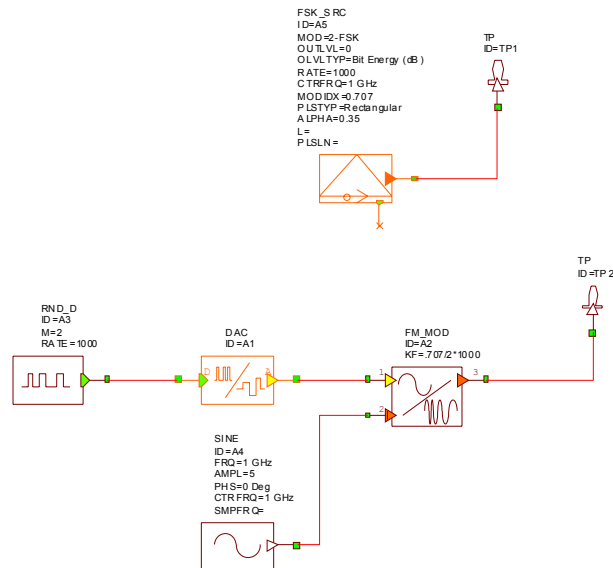
- 7 Run the simulation. The power spectrum of the FSK signal generated by the block displays as shown in the following graph.



## Creating An FSK Modulator Using Elementary Blocks

In this section you create a BFSK modulator using elementary blocks, then compare the results with those of the previous method. These two distinct transmitters are behaviorally identical.

- 1 In the Element Browser, expand the **Sources** category, then click the **Random** subgroup. Select the RND\_D block and place it on the system diagram as shown in the following figure.
- 2 Double-click the RND\_D block and set the RATE parameter to “**1000**” and the RSEED parameter to “**{0}**” (with the brackets).
- 3 Expand the **Converters** category, then click the **Analog-Digital** subgroup. Select the DAC block and place it on the system diagram as shown in the following figure.
- 4 Expand the **Modulation** category, then click the **Analog** subgroup. Select the FM\_MOD block and place it on the system diagram as shown in the following figure.
- 5 Set the FM\_MOD block KF parameter to “**0.707/2\*1000**”.
- 6 Expand the **Sources** category, then click the **Waveforms** subgroup. Select the SINE block and place it on the system as shown in the following figure.
- 7 Set the SINE block AMPL parameter to “**5**”. Leave the FRQ parameter at 1GHz.
- 8 Add a second test point to the system as shown in the following figure.



- 9 Add a power spectrum (PWR\_SPEC) measurement at test point **TP2** to the “Spectrum” graph.
- 10 Run the simulation. Note that there is no change in the graph since the waveform from the cascade of these blocks is identical to the waveform of the FSK\_SRC block, hence the overlap.

**TIP:**

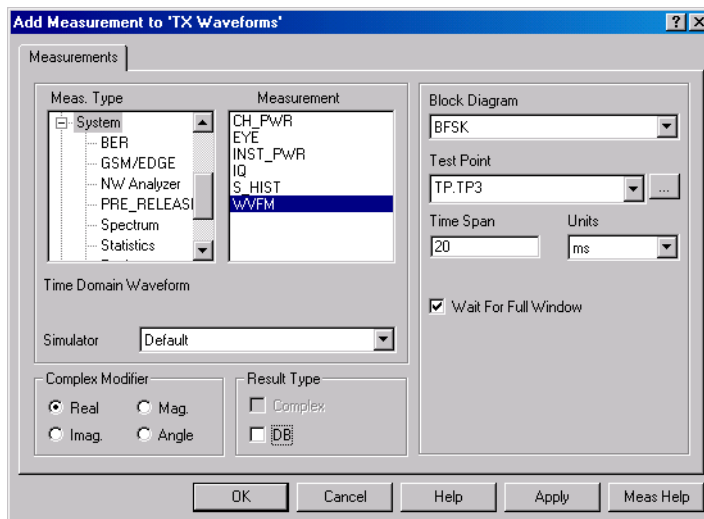
To view the spectrum of each setup separately, you can display them in separate graphs or toggle the measurements one at a time under **Graphs** in the Project Browser (right-click and choose **Toggle Enable**). You can also simply toggle the test points one at a time by double-clicking the test point and clearing the **Enable element** check box in the Element Options dialog box. These steps confirm that the two methods for setting up the transmitter create identical BFSK waveforms.

- 11 Add a third test point between the RND\_D block and the DAC block.

To observe the behavior of the transmitted phase:

- 12 Add a rectangular graph named “**TX Waveforms**”.

- 13 Add a random data waveform measurement to the graph using the settings in the following figure, then click **Apply**.

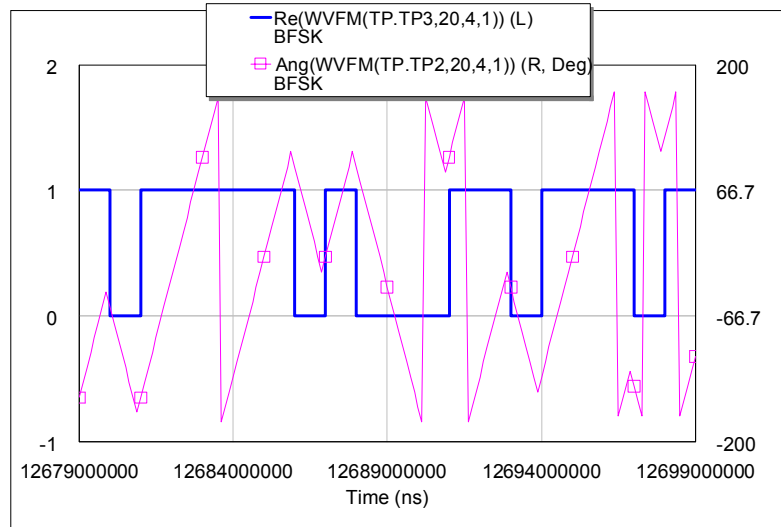


- 14 Add a measurement to the graph for the phase produced by the FSK modulator using the settings in step 13, but select **TP.TP2** as the **Test Point** and **Angle** as the **Complex Modifier**, then click **OK**.
- 15 Select the “TX Waveforms” graph and click the **Properties** button on the toolbar or right click the graph and select **Properties**.
- 16 Click the **Axes** tab and select **Left 1**. Clear the **Auto limits** check box and enter “-1” as the **Min** and “2” as the **Max**. Under **Divisions**, clear the **Auto divs.** check box and enter “1” as the **Step**.
- 17 Select **Right 1**. Clear the **Auto limits** check box and enter “-200” as the **Min** and “200” as the **Max**, then click **Apply**.
- 18 Click the **Measurements** tab and under **Select Measurement to edit**, select **BFSK:Ang(WVFM(TP.TP2,20,4,1))**, then under **Choose axis**, select **Right 1** and click **Apply**.
- 19 Click the **Traces** tab and under **Style**, select measurement 1, then under **Weight** select a heavier line from the corresponding drop-down box at the bottom of the dialog box. Select measurement 2, then select a square as the **Symbol** style from the corresponding drop-down box at the bottom of the dialog box.



- 20 Run the simulation. Observe that the transmitted phase behaves exactly as expected in a binary FSK transmission scheme with rectangular frequency shaping pulse.

Your simulation response should look similar to the following graph, which shows that the phase of the modulated waveform increases in a ramp when the input bit is “1”, and decreases in a ramp with the same slope when the input bit is “0”. The phase remains continuous between different bit intervals, and the phase jumps in the plot are only the effect of the wrap-around when the phase exceeds  $\pm\pi$ . The data waveform (binary 1’s and 0’s) is plotted on the left axis while the phase waveform is shown on the right axis.

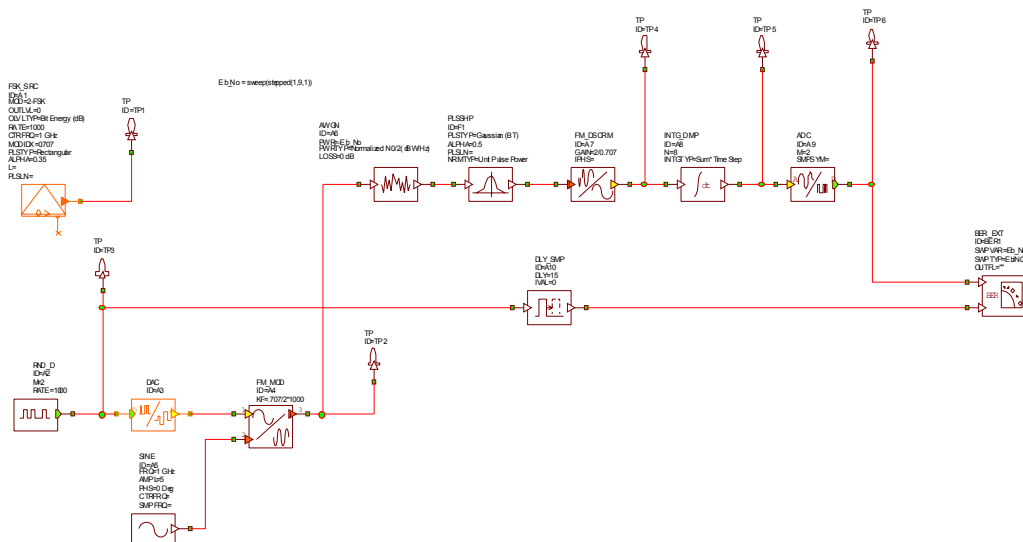


## Receiver and Demodulation

In this section you complete the channel-receiver chain for BFSK. The receiver is built from elementary VSS blocks for demonstration purposes and is shown to outperform even the coherent BFSK demodulator. The modulated signal out of the transmitter is passed through an AWGN channel and then demodulated by a discriminator receiver, consisting of a filter (PLSSHP), an FM discriminator (FM\_DSCRM), an integrate-and-dump block (INTG\_DMP), and an ADC block. In this exercise you also scale the channel and noise parameters and analyze the system performance.

To complete the channel-receiver chain for BFSK (using the following figure as a reference):

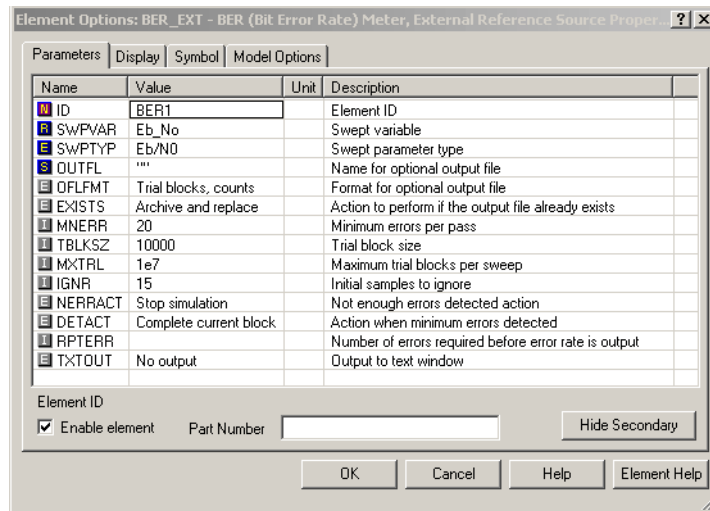
- 1 In the Element Browser, click the **Channels** category. Select the AWGN block and place it on the system diagram.
- 2 Click the **Filters** category, then select the PLSSHP block and place it on the system diagram.
- 3 Expand the **Modulation** category, then click the **Analog** subgroup. Select the FM\_DSCRM block and place it on the system diagram.
- 4 Click the **Signal Processing** category, then select the INTG\_DMP block and place it on the system diagram.
- 5 Expand the **Converters** category, then click the **Analog-Digital** subgroup. Select the ADC block and place it on the system diagram.
- 6 Add a 4th, 5th and 6th test point at the outputs of the FM\_DSCRM, INTG\_DMP and ADC blocks respectively.
- 7 Click the **Signal Processing** category, then select the DLY\_SMP block and place it on the system diagram.
- 8 Expand the **Meters** category, then click the **BER** subgroup. Select the BER\_EXT block and place it on the system diagram.



- 9 Add the following equation to the system diagram.

$$Eb\_No = \text{sweep}(\text{stepped}(1, 9, 1))$$

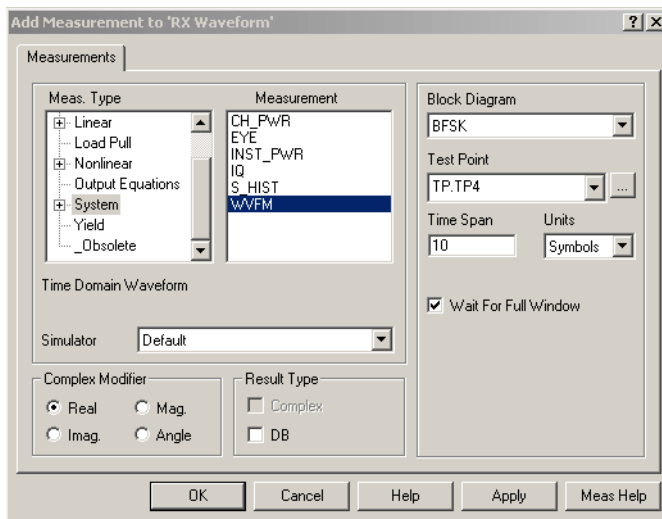
- 10 Set the AWGN block PWR parameter to “-Eb\_No” and its PWRTYP parameter to “**Normalized N0/2(dBW/Hz)**”.
- 11 Set the PLSSHP block PLSTYP parameter to “**Gaussian (BT)**” and its ALPHA parameter to “**0.5**”.
- 12 Set the FM\_DSCRM block GAIN parameter to “**2/0.707**”.
- 13 Set the INTG\_DMP block N parameter to “**8**”.
- 14 Set the ADC block M parameter to “**2**”.
- 15 Set the DLY\_SMP block DLY parameter to “**15**”.
- 16 Verify that the BER\_EXT block parameters match those in the following figure.



## Adding Graphs and Analyzing Results

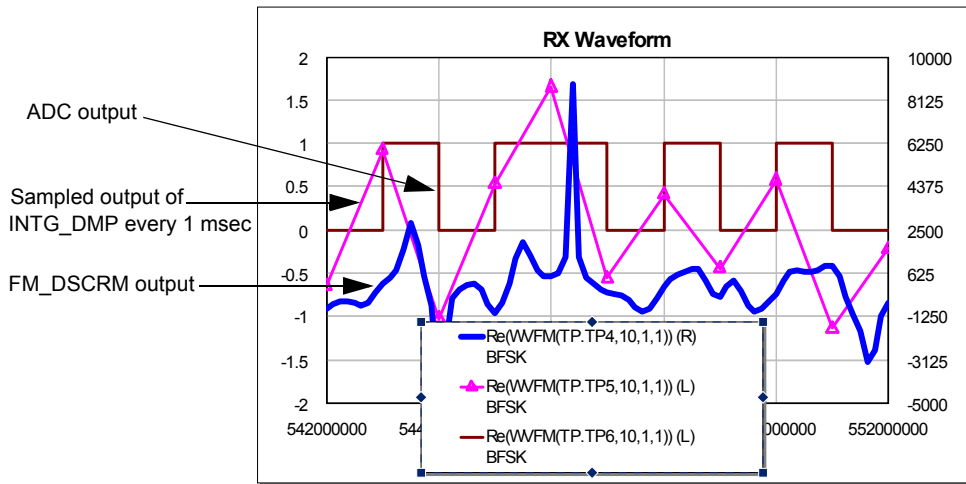
To add and analyze an RX waveform graph:

- 1 Add a rectangular graph named “**RX Waveforms**”.
- 2 Add a measurement to this graph using the following settings, then click **Apply**.

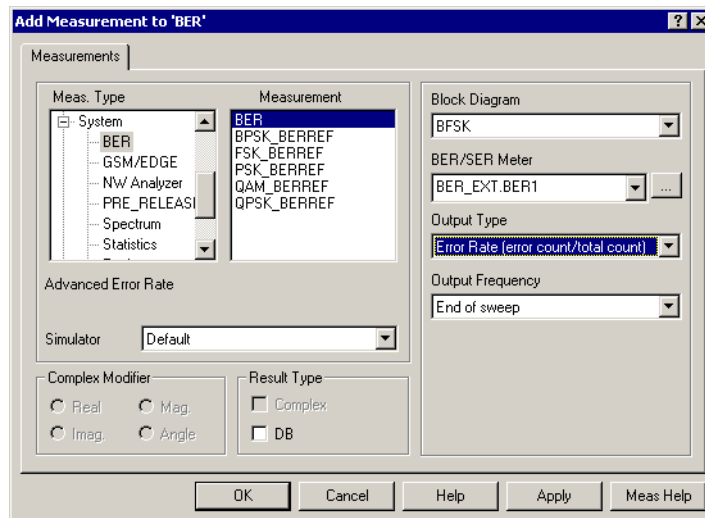


- 3 Repeat step 2 but select **TP.TP5** as **Test Point**.
- 4 Repeat step 2 but select **TP.TP6** as **Test Point**, then click **OK**.
- 5 Run the System Simulator for a few seconds and then stop. A snapshot of the simulator response is shown in the following graph. Your graph should look very similar after making the appropriate trace and axis property changes in the Graph Properties dialog box. Set the waveform at testpoint TP.TP4 to display on the right axis.
- 6 Analyze the graph. The bold waveform is the output of the FM discriminator in low noise. The sample output of the integrate-and-dump block at every 1 msec or 1e6 nsec (sample at the symbol rate of 1000 samples/sec) is displayed with a triangular mark. As expected, this curve is positive when the FM discriminator output is increasing, and negative otherwise. Also observe the output of the ADC block, which slices the output of the integrate-and-dump block for digital output data. The scale for integrate-and-dump output and ADC output is on the left y-axis, and that of the FM discriminator is on the right y-axis.

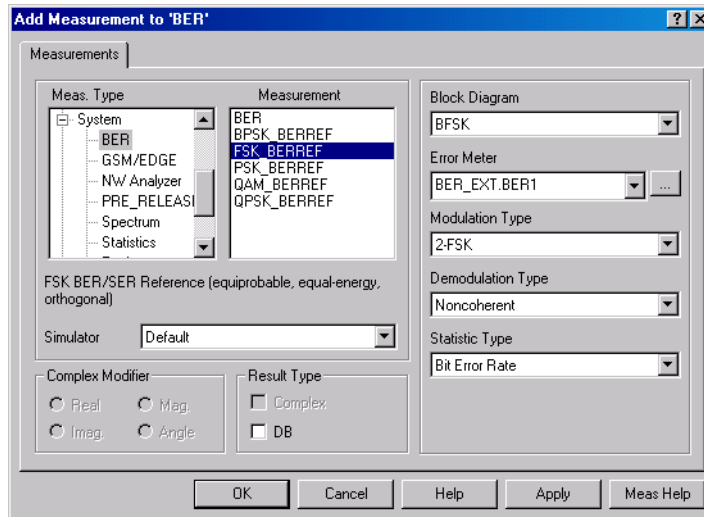




- 7 Add another rectangular graph named “BER”.
- 8 Add a measurement to this graph using the settings in the following figure, then click **Apply**.

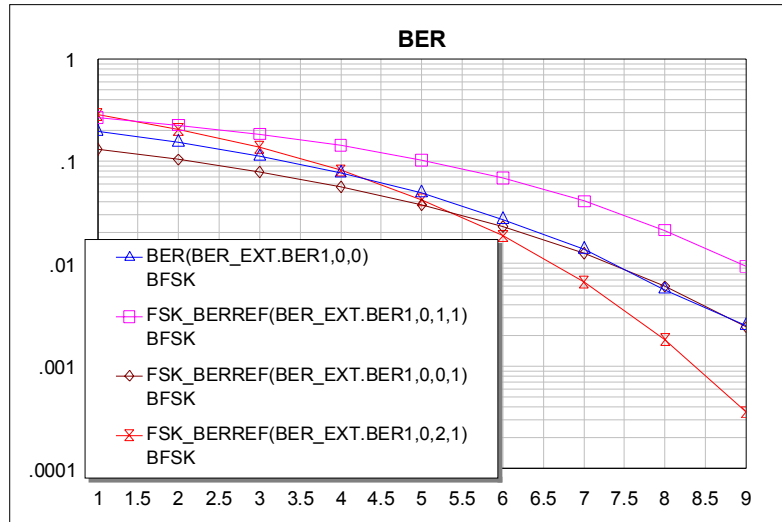


- 9 Add another measurement to this graph to represent the performance of a non-coherent receiver, using the settings in the following figure, then click **Apply**.



- 10 Repeat step 9 but select “**Coherent**” as **Demodulation Type**. This represents the performance of a coherent linear (correlation) receiver.
- 11 Repeat step 9 but select “**Discriminator**” as **Demodulation Type**, then click **OK**. This represents the performance of the nonlinear discrimination receiver, under an ideal assumption.
- 12 Set the BER\_EXT block TXTOUT parameter to “**Trial statistics**”.
- 13 Run the simulation. A text window displays with the statistics of BER simulation.
- 14 Select the BER graph window, then click the **Properties** button on the toolbar. Click the **Traces** and **Axes** tabs and make the appropriate changes to your graph so it looks similar to the following graph.





- 15 Set the SINE block AMPL parameter to " $5 \cdot 10^{(Eb\_No/20)}$ ".
- 16 Set the AWGN block PWR parameter to "0" db.
- 17 Run the simulation again. Note that these changes have no effect on the BER simulation; the results are identical. If you use the FSK\_SRC block in the system (instead of elementary blocks), you can achieve the identical BER simulation by setting its OUTLVL parameter to "0" db and the AWGN block PWR parameter to " $-Eb\_No$ ", or by setting OUTLVL to " $Eb\_No$ " and PWR to "0" db.
- 18 Save and close the project.

## I/Q IMBALANCE EXAMPLE

In this section you observe the effect of amplitude and/or phase mismatch (I/Q imbalance) of a complex signal. For more information, see the IQ Imbalance application notes available in the KnowledgeBase on AWR's website at [www.awrwave.com](http://www.awrwave.com).

The procedure in this example analyzes the effect of I/Q imbalance in a QAM signal. The complete example is available as "IQ\_Imbalance.emp" in your

*Program Files\AWR\AWR2004\Examples\Getting Started\VSS\IQ\_Imbalance* directory.

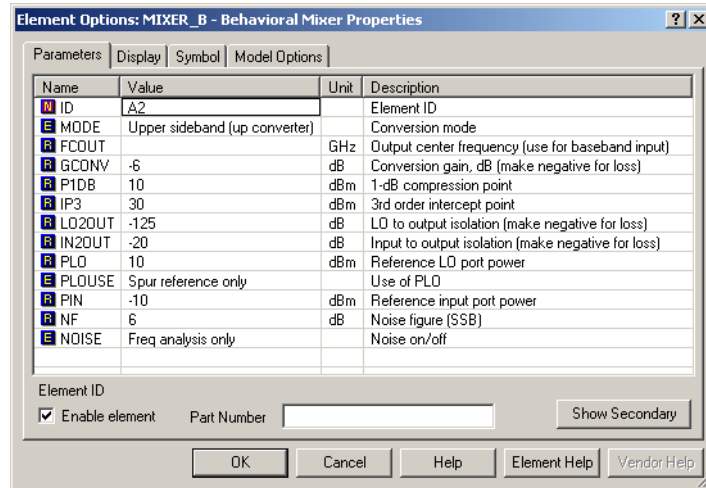
## I/Q Imbalance and Image Rejection Ratio

Any digital transmitter-receiver communication system which comprises analog and digital sections is plagued by “in-phase to quadrature-phase” (I-Q) imbalance, causing the signal to distort. I-Q imbalance occurs when the quadrature-phase signal components of the modulated signal are not perfectly in quadrature (separated in-phase by 90-degrees), or are otherwise processed unequally, such as the application of differing gain to in-phase and quadrature signals when equal gain is desired. I-Q imbalances typically occur at least in the analog sections of the communication system, particularly in connection with upconversion and downconversion. In this section you study the I-Q imbalance in a QAM signal by the effect of either a dirty input signal or a dirty local oscillator (LO). You also measure the image rejection ratio (IRR).

To view the I-Q imbalance in a QAM signal:

- 1 Create a new project named “IQ\_Imbalance” and add a system diagram named “**QAM System**”.
- 2 In the Element Browser, expand the **XML Libraries** category, then expand the **RF Blocks** subgroup. Click the **Imbalance** group, then select the INP\_IMBAL block and place/connect it on the “QAM System” system diagram as shown in the following figure.
- 3 Expand the **Modulation** category, then click the **QAM** subgroup. Select the QAM\_SRC block and place it on the system diagram as shown in the following figure. Set the MOD parameter to “**16-QAM (Gray)**”, the OUTLVL parameter to “**10**”, the OLVLTY parameter to **Avg. Power (dBm)**, and the PLSTYP parameter to **Raised Cosine**.
- 4 Expand the **RF Blocks** category, then click the **Mixers** subgroup. Add a MIXER\_B block with parameters set as shown in the following system diagram.



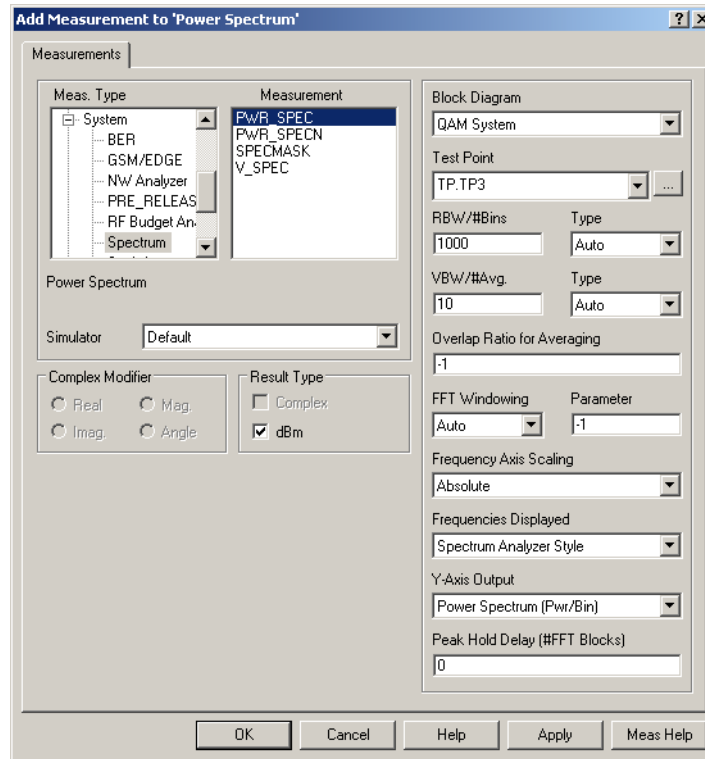


- 5 Expand the **RF Blocks** category, then click the **Sources** subgroup. Select the TONE block and place it on the system diagram as shown on the following system diagram.
- 6 Set the TONE block FRQ parameter to **5.2 GHz**.
- 7 Click the **Signal Processing** category, select the DECIM block and place two of them on the system diagram as shown in the following system diagram.
- 8 Set the DECIM block RATIO parameter to “**8**” and the DLY parameter to “**4**” for both blocks.
- 9 Add four test points to complete the system as shown in the following system diagram.

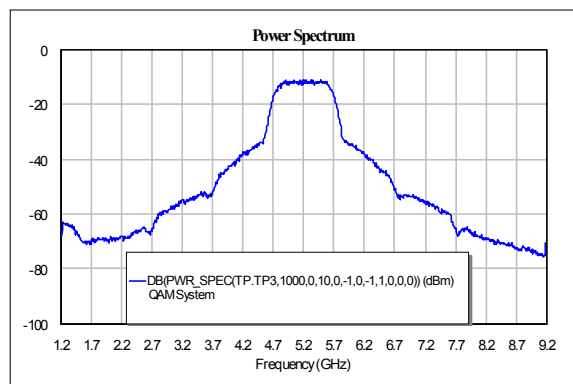
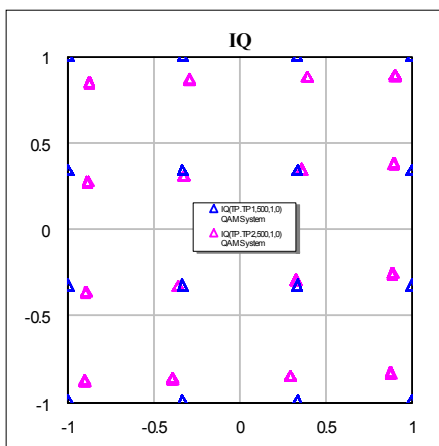


To add and analyze a constellation graph:

- 13-16** MWO/VSS 2004



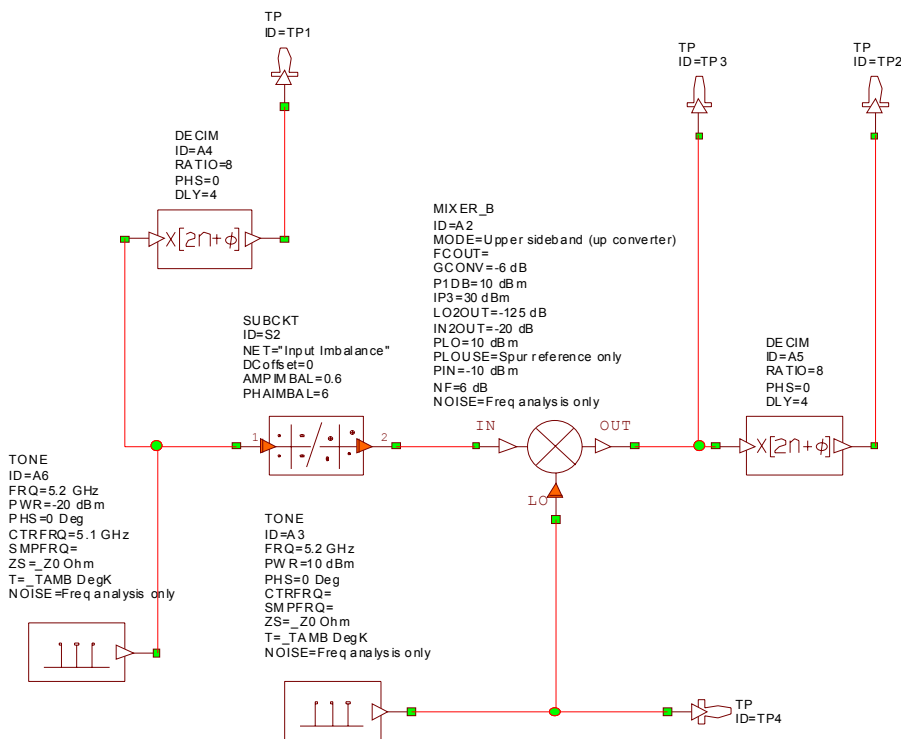
- 5 Run the simulation. The simulation responses shown in the following graphs should display.



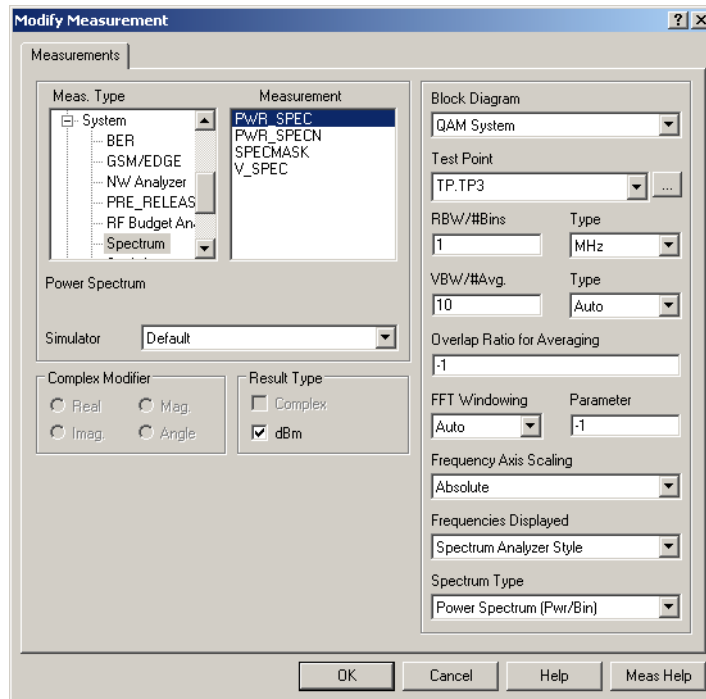
The constellation at the output is slightly skewed. You can alter the AMPIMBAL and PHAIMBAL parameters and analyze the effect on the constellation even further. The system thus simulated has a dirty input signal (causing imbalance) and a clean LO. You can also build the system with a clean input signal and a dirty LO (causing imbalance). To do so you need to remove the INP\_IMBAL block and add the LO\_IMBAL block between the TONE block and the “LO” terminal of the MIXER\_B block. This also causes the output constellation to skew in a similar pattern.

The INP\_IMBAL block has an amplitude imbalance of 0.6, a phase imbalance of 6 degrees and a DC offset value of  $20 \times 10^{-3}$  (or 0.02). If the DC offset value is zero, ideally it should produce an image rejection ratio (IRR) of 24 dB. Here, you display the IRR for tones

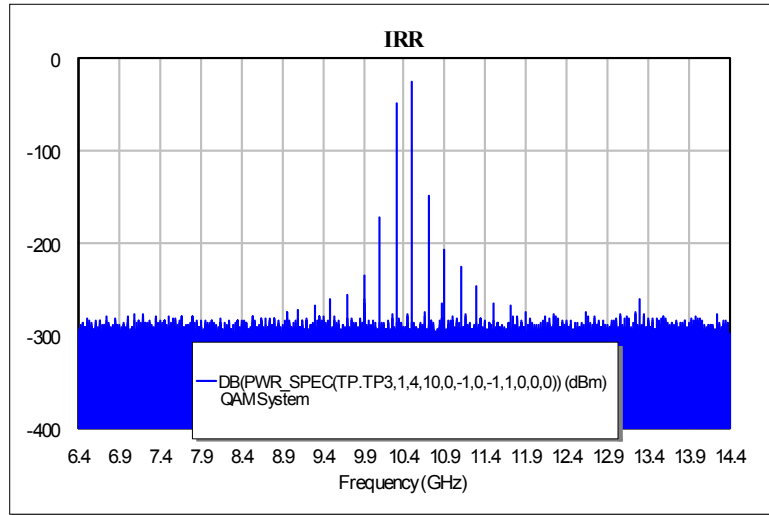
- 6 Set the DCOffset parameter of the Input Imbalance subcircuit to “0”.
- 7 Replace the QAM\_SRC block with a copy of the TONE block as shown in the following figure. Set the new TONE block FRQ parameter to “5.2” GHz and the CTRFRQ parameter to “5.1” GHz.



- 8 Add a rectangular graph named “**IRR**”.
- 9 Add a measurement to the graph using the settings in the following figure, then click **OK**.



- 10 Run the simulation. The simulation response in the following graph should display. Measure the difference between the power spectrums of two IF frequencies, it is almost 24 dB.



11 Save and close the project.

## END-TO-END 64QAM EXAMPLE

This example presents an advanced end-to-end 64 QAM system. You build the system comprising an RF chain, and observe the error vector magnitude (EVM), adjacent channel power ratio (ACPR), AM-to-AM characteristics, and intersymbol interference (ISI) introduced by the filter.

The procedures in this section include:

- Building the QAM system
- Analyzing the non-linearities of an amplifier
- Analyzing ISI in the system.

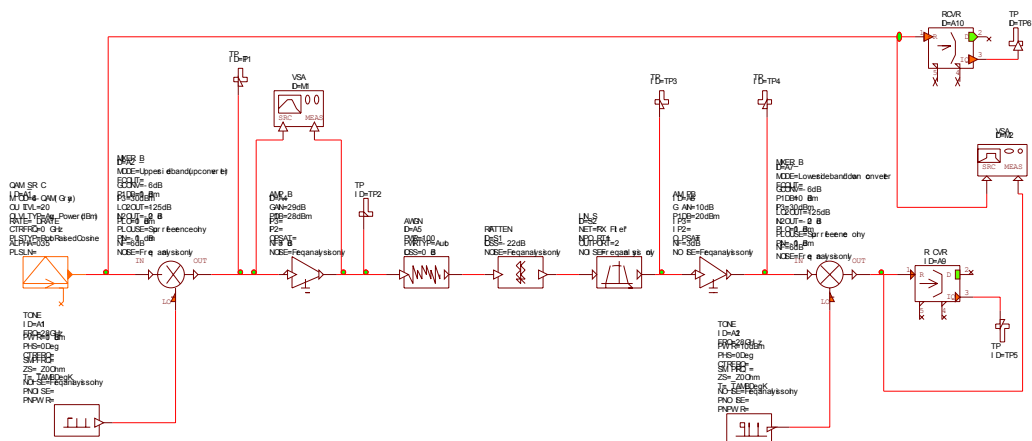
## Building a QAM System

The example you create in this section is available as “64QAM.emp” in your *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\64QAM* directory.

To build a QAM system:

- 1 Create a project named “**64QAM**”.
- 2 Add a system diagram named “**64QAM System**”.
- 3 To set the project options choose **Options > Project Options** and click the **Global Units** tab. Set the **Frequency** as **GHz**, **Time** as **ns** and **Power/Log** as **dBm**.
- 4 Choose **Options > Default System Options** and click the **Simulator** tab. Select **Data rate (\_DRATE)** and set its value to “**0.005**” GHz, then select **Oversampling (\_SMPSYM)** and set its value to “**8**” samples/symbol.
- 5 Right-click **Circuit Schematics** and import the “RX Filter.sch” file located in the *Program Files\AWR\AWR2004\Examples\Getting Started\VSS\64QAM* directory.
- 6 Expand the **Modulation** category, then click the **QAM** subgroup. Select the QAM\_SRC block and place it on the system diagram as shown in the following figure. Set the MOD parameter to “**64-QAM (Gray)**” and the OUTLVL parameter to “**-20**” dBm.
- 7 Expand the **RF Blocks** category, then click the **Mixers** subgroup. Select the MIXER\_B block and place/connect it on the system diagram as shown in the following figure.
- 8 Select the TONE block and place it on the system diagram as shown in the following figure. Set the FRQ parameter to “**28**” GHz. Connect the output of the TONE source to the LO input of MIXER\_B. Set the MODE parameter to “Upper sideband (up converter)”
- 9 Expand the **RF Blocks** category, then click the **Amplifier** subgroup. Select the AMP\_B block and place/connect it on the system diagram as shown in the following figure. Set the GAIN parameter to “**29**”, and the P1DB parameter to “**28**”.
- 10 Click the **Channels** category, then select the AWGN block and place/connect it on the system diagram as shown in the following figure. Set the PWR parameter to “**-100**”.

- 11 Expand the **RF Blocks** category, then click the **Other** subgroup. Select the RFATTEN block and place/connect it on the system diagram as shown in the following figure. Set the LOSS parameter to “-22”.
- 12 Expand the **Linear Filters** category, then click the **Simulation Based** subgroup. Select the LIN\_S block and place/connect it on the system diagram as shown in the following figure. Set the NET parameter to “**RX Filter**”.
- 13 Add another AMP\_B block, MIXER\_B block, and TONE block and connect the LO nodes of the two mixers as shown in the following figure. Ensure that MIXER\_B is set the MODE to downconvert, and that the FRQ parameter of the TONE block is set to “**28**” GHz.
- 14 Set the copy of the AMP\_B block GAIN parameter to “**10**”, and the P1DB parameter to “**20**”.
- 15 Expand the **Modulation** category, then click the **General Receivers** subgroup. Select and place/connect two RCVR blocks on the system diagram as shown in the following figure.
- 16 Expand the **Meters** category, then click the **Network Analyzer** subgroup. Select and place/connect two VSA blocks on the system diagram as shown in the following figure.
- 17 Add six test points in the same order as shown on the system diagram in the following figure.





- 18 Set both MIXER\_B block parameters (except the IDs and MODE) to the values shown in the following figure, then click **OK**.

**Element Options: MIXER\_B - Behavioral Mixer Properties**

Parameters | Display | Symbol | Model Options

Name	Value	Unit	Description
ID	A2		Element ID
MODE	Upper sideband (up converter)		Conversion mode
FCOUT		GHz	Output center frequency (use for baseband input)
GCONV	-6	dB	Conversion gain, dB (make negative for loss)
P1DB	10	dBm	1-dB compression point
P1DBTYP	Input P1dB		P1dB interpretation
IP3	30	dBm	3rd order intercept point
IP3TYP	Input IP3		IP3 interpretation
LO2OUT	-125	dB	LO to output isolation (make negative for loss)
LO2OUTH		dB	LO to output harmonics (>1) isolation (make negative for loss)
IN2OUT	-20	dB	Input to output isolation (make negative for loss)
IN2OUTH		dB	Input to output harmonics (>1) isolation (make negative for loss)
PLO	10	dBm	Reference LO port power
PLOUSE	Spur reference only		Use of PLO
PIN	-10	dBm	Reference input port power
NF	6	dB	Noise figure (SSB)
NOISE	Freq analysis only		Noise on/off
BBOP	Auto		Baseband operation
LOHMAX	1		Maximum LO harmonic to generate if IMPROD is empty
INHMAX	1		Maximum input harmonic to generate if IMPROD is empty
IMPROD			Harmonics and intermodulation products to generate (vector...)
SIGBW	100		Signal bandwidth as percentage of sampling frequency
SPURBW	100		Spur threshold bandwidth as percentage of sampling frequency
AALIAS	Auto		Anti-aliasing handling
MXUPSMP	1000		Maximum upsampling rate for anti-aliasing
TXTOUT	None		Output to text window

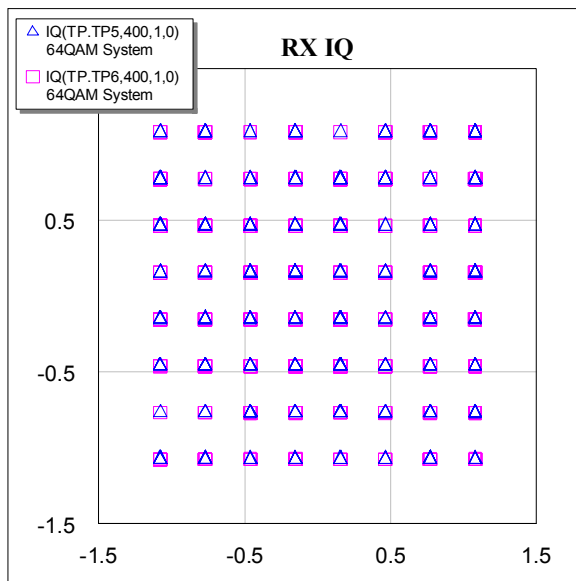
Element ID  
☒ Enable element    Part Number    

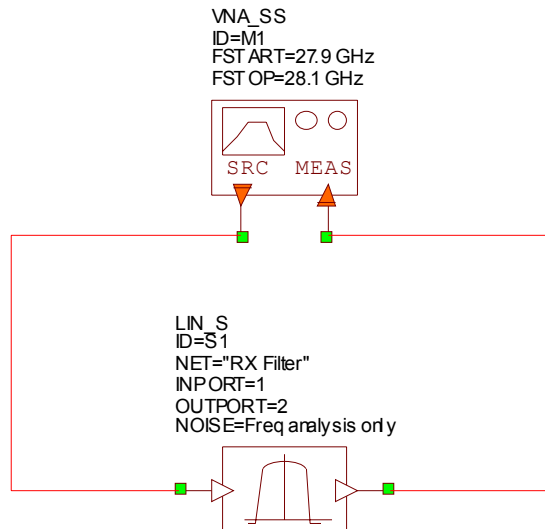
## Adding Graphs and Analyzing Results

To add and analyze a constellation and rectangular graph:

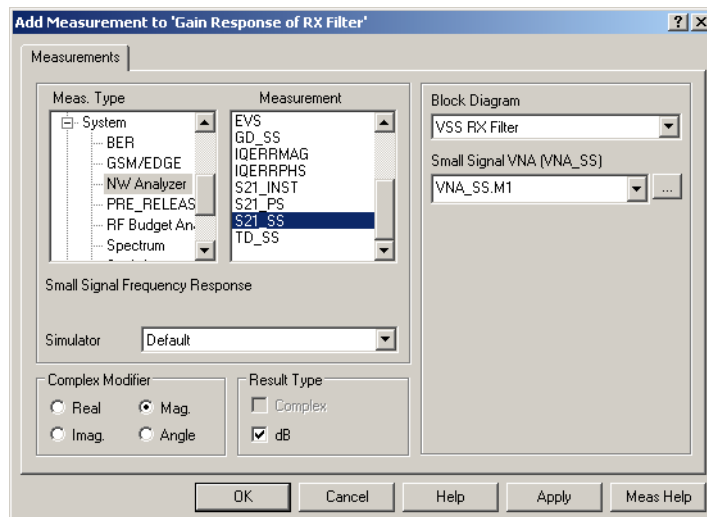
- 1 Add a constellation graph named “**RX IQ**”.
- 2 Add two System/IQ measurements to the graph to display the constellations at test points TP6 and TP5 (the output of the RCVR blocks). Set the **Block Diagram** to **64QAM System**, the **Time Span** to “**400**”, and the **Units** to **Symbols**. Make sure you use a different symbol for TP6 and TP5.
- 3 Run the simulation. The simulation response in the following graph should display. You can clearly see the effect of inter-symbol interference (ISI) introduced by the filter on the received signal.



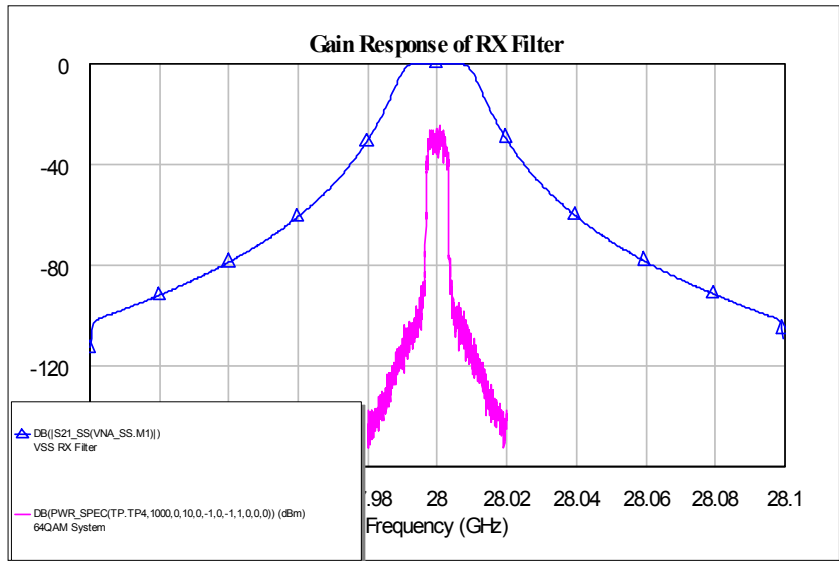
- 4 Disable and bypass the LIN\_S block, then run the simulation again. Notice the difference in the signal constellation-- the ISI is minimized. Disabling the AWGN block would further reduce the jitter in the system.
- 5 Create a system diagram named **“VSS RX Filter”**.
- 6 Copy the LIN\_S block from the “64QAM System” system diagram and place it in the “VSS RX Filter” system diagram.
- 7 Expand the Meters category and click the Network Analyzers subgroup. Select a VNA\_SS block and place/connect it in the “VSS RX Filter” system diagram as shown in the following figure.



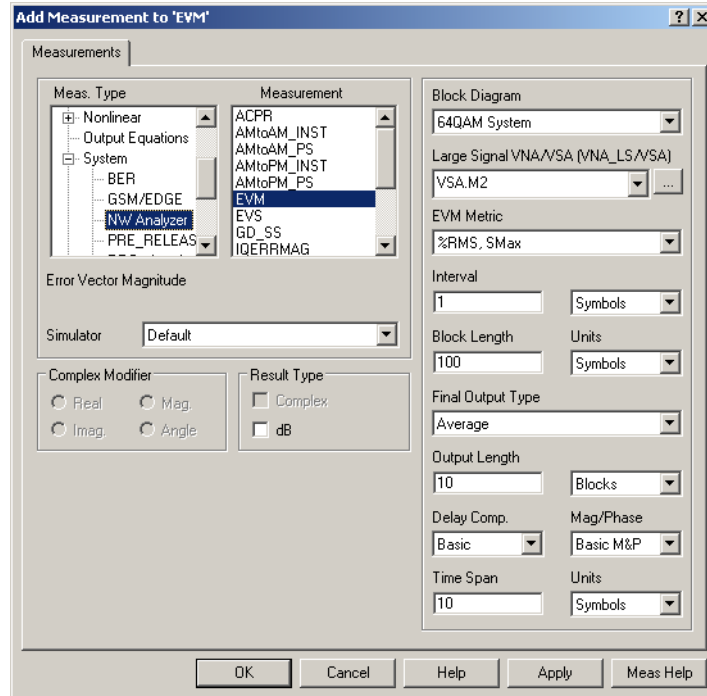
- 8 In the VNA\_SS block, set the FSTART parameter to “27.9” and the FSTOP parameter to “28.1”.
- 9 Add a rectangular graph named “**Gain Response RX Filter**”.
- 10 Add a measurement to the graph using the settings in the following figure, then click **OK**. (Make sure the **dB** check box is selected.)



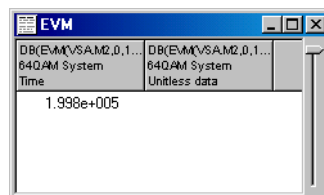
- 11 Add a Power Spectrum measurement to the “Gain Response RX Filter” graph from the **System > Spectrum** group (use default parameter settings) at Test Point 4.
- 12 Run the simulation. The simulation response in the following plot should display.



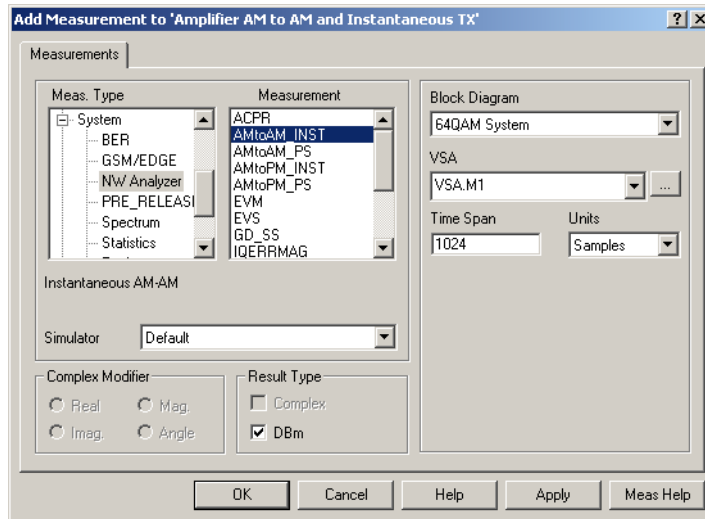
- 13 Add a tabular graph named “**EVM**” and add a measurement using the following settings, then click **OK**.



Note that the VSA block selected in **VSA/Large Signal VNA** should be the block with the QAM\_SRC and mixer connected to it (VSA.M2). Also note that the EVM measurement auto-compensates for gain, phase, and delay. See the online Help for this measurement for more information. Simulate and read the EVM from the tabular graph.



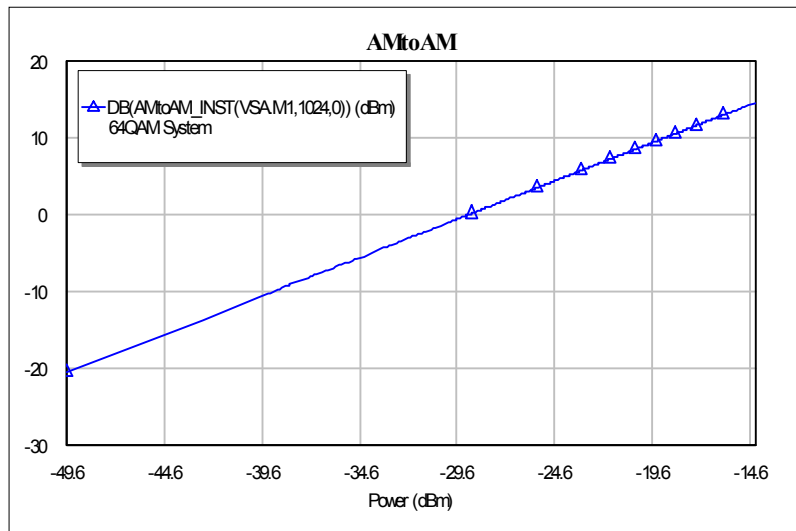
- 14 Add a rectangular graph named “**AM to AM**”, and using the settings in the following figure, add a measurement to analyze the instantaneous AM-to-AM characteristic of the signal, then click **OK**.



Note that the VSA block selected in **VSA** should be the block connected across the AMP\_B block. Your block ID may differ from the **VSA.M1** shown in this dialog box.



- 15 Run the harmonic balance simulator and system simulator. The simulation response in the following graph should display.



- 16 Tune the **OUTLVL** parameter of the QAM\_SRC block and view the impact on the constellation graph.

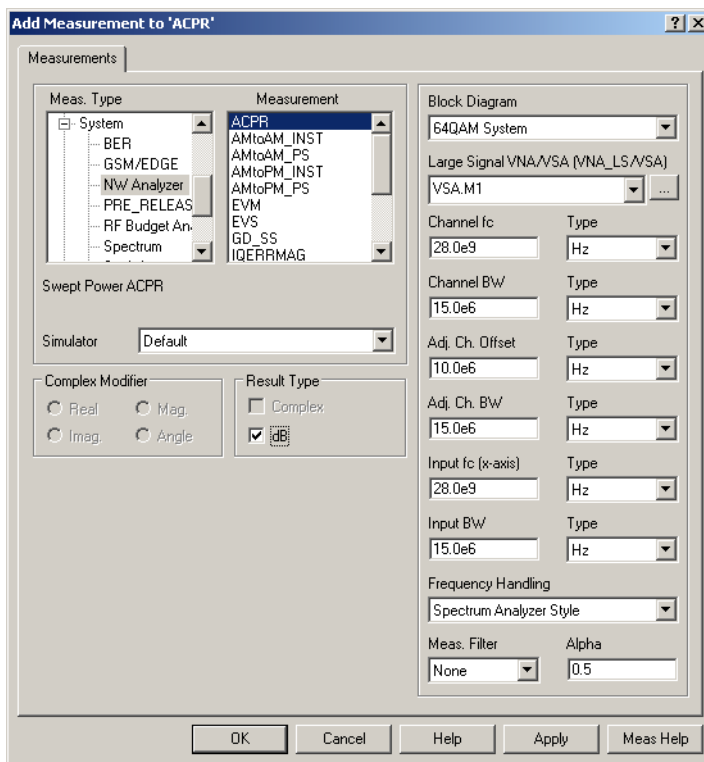
## Adjacent-Channel Power Ratio (ACPR)

In this section you observe the adjacent-channel power ratio imposed by the nonlinear effect of the power amplifier. You can calculate ACPR in two different ways. The first is by finding  $10 \cdot \log$  of the ratio of the total output power to the power in the adjacent channel. The second and most popular method, shown here, is finding the ratio of the output power in a smaller bandwidth around the center of a carrier, to the power in the adjacent channel. The smaller bandwidth is equal to the bandwidth of the adjacent channel signal. In VSS you can measure the ACPR at a given signal power or over a range of signal power.

In the following simple example, the channel of interest is centered at 28 GHz and has a bandwidth of 15 MHz. The adjacent channel is centered at 28.010 GHz and also has a bandwidth of 15 MHz. The input carrier signal is centered at 28 GHz and has a bandwidth of 15 MHz.

To measure the ACPR at the signal power of -20 dBm:

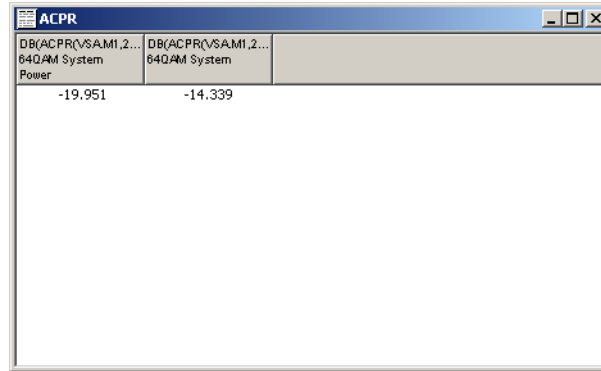
- 1 Add a tabular graph named “**ACPR**” and add a measurement using the following settings, then click **OK**.



Note that the VSA block selected in **Vector Signal Analyzer** should be the block connected across the AMP\_B block. Your block ID may differ from the **VSA.M1** shown in this dialog box.

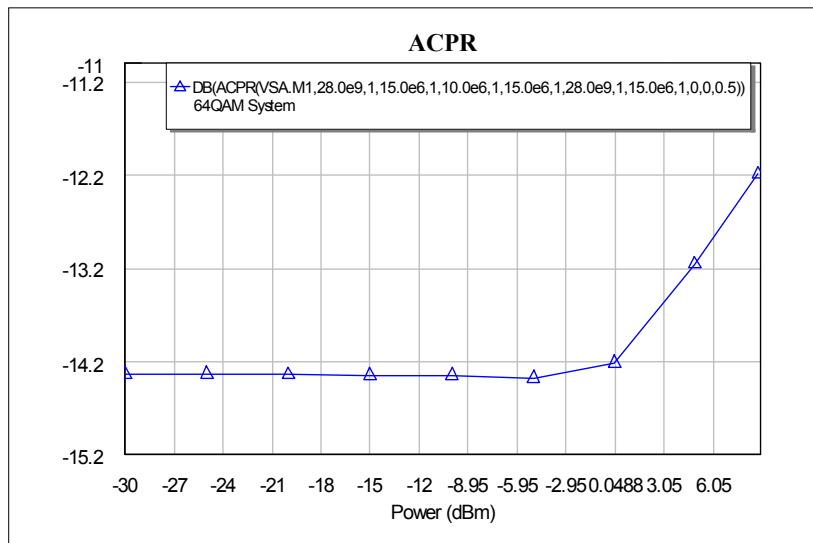
- In the VSA.M1 block specified above, set the SWPDUR parameter to “10” ns.
- Start the simulation. The tabulated form displays the ACPR value shown in the following table.





To measure the ACPR over a range of signal power:

- 1 In the QAM\_SRC block, set the OUTLVL parameter to “**sweep(stepped(-30,10,5))**” without the quotes.
- 2 Start the simulation again and let it run for a couple of minutes. Notice how the ACPR tabular graph updates. You can display the results in a rectangular graph by right-clicking **ACPR** under **Graphs** in the Project Browser and choosing **Change Type to > Rectangular**. The simulation response in the following graph should display.



- 3 Save and close the project.

## MIXER MODELING EXAMPLE

---

In this example you model a mixer from a given set of intermodulation and harmonic suppression (spurs) data. The spur suppressions for the mixer are specified through a text data file. You can use different text data file formats to specify the spurs.

Here you use the MIXER\_F block which implements a behavioral model of the nonlinear mixer. This block is normally used to model a specific mixer only when the spur table is available, otherwise you should use the MIXER\_B block instead.

This procedures in this section include:

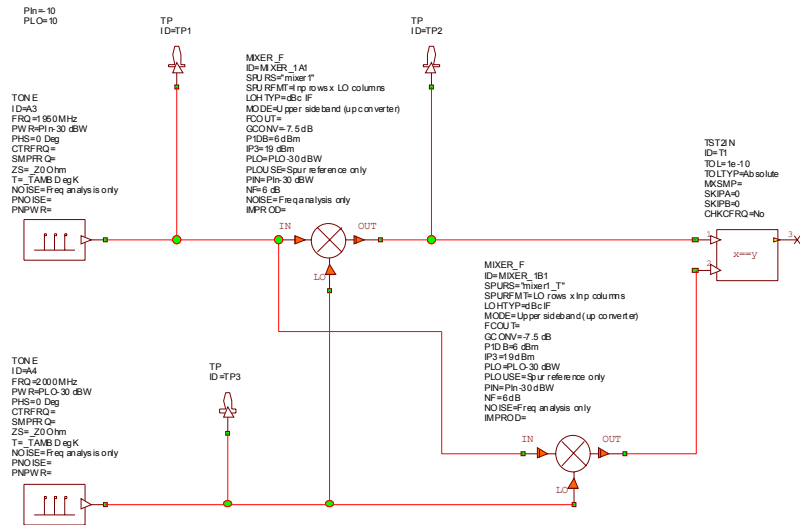
- Modeling a mixer
- Using various data file formats and verifying the results

## Setting up the System

The example you create in this section is available in its complete form as MIXER.emp. This file is included in your *ProgramFiles\AWR\AWR2004\Examples\Getting Started\VSS\MIXER* directory. In this example you model the mixers using a given data file and its transpose file and then verify that the two instances are identical.

To build a mixer system:

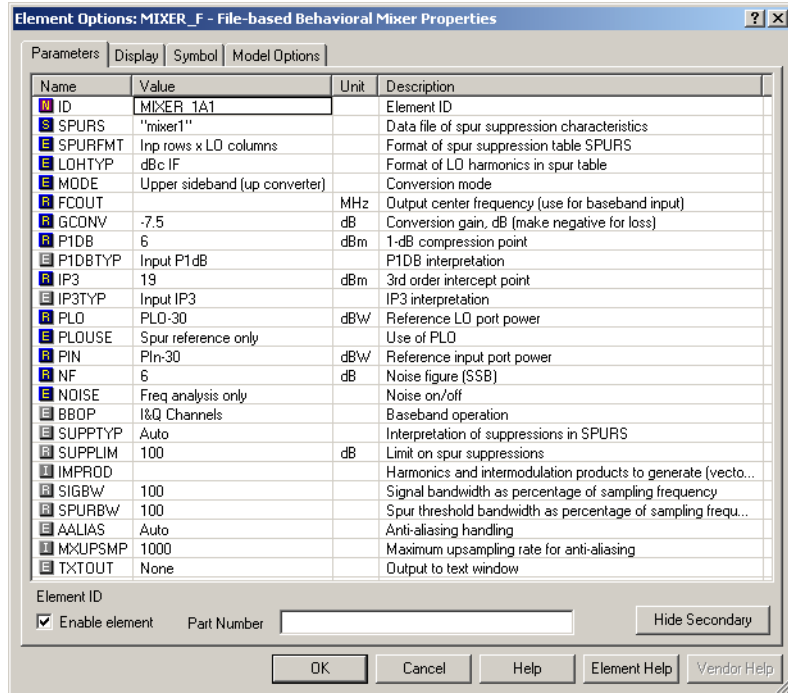
- 1 Create a project named “**Mixer**”.
- 2 Choose **Options > Project Options**, click the **Global Units** tab, and ensure that **Frequency** is set to “**MHz**”.
- 3 Import the “Mixer1.sys” system diagram located in the *ProgramFiles\AWR\AWR2004\Examples\Getting Started\VSS\MIXER* directory. “Mixer1” is shown in the following diagram.



- 4 Import the “mixer1.txt” and “mixer1\_T.txt” data files located in the *ProgramFiles\AWR\AWR2004\Examples\Getting Started\VSS\MIXER* directory. In the “mixer1” data file, each row represents the input harmonic and each column represents the LO harmonic, with the top row and left column representing the first harmonic. The “mixer1\_T” data file is the transpose of the “mixer1” data file with rows representing LO harmonics and columns representing the input harmonic.

On the “Mixer1” system diagram:

- 5 Depending on the type of data file used, you must set the SPURFMT parameter accordingly. Verify that the MIXER\_1A1 block has the following parameter values.



- 6 In the MIXER\_1B1 block, ensure that SPURS is set to **"mixer1\_T"**, SPURFMT to **"LO rows x Inp columns"**, and set the remaining parameters to the MIXER\_1A1 values as shown in the previous figure.

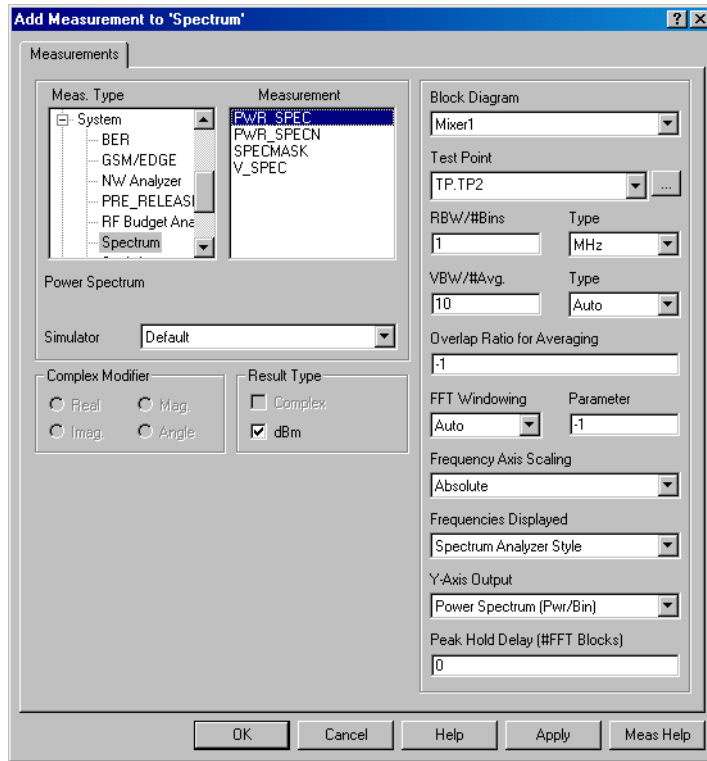
## Adding Graphs and Analyzing Results

With FLO=2000MHz, FIn=1950MHz, PLO=+10dBm, PIn=-10dBm, GCONV=-7.5dB, the expected tones are shown in the following figure.

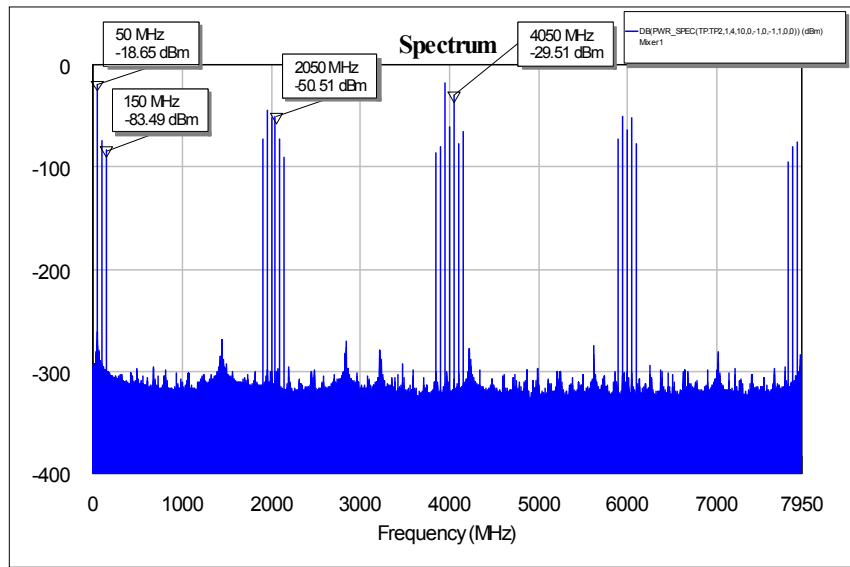
LO (n)	RF (m)	MHz	dBc	dBm
1	-1	50	0	-17.5
2	-2	100	56	-73.5
3	-3	150	68	-85.5*
-1	2	1900	55	-72.5
0	1	1950	27	-44.5
1	0	2000	34	-51.5
2	-1	2050	33	-50.5
3	-2	2100	55	-72.5
4	-3	2150	73	-90.5
-1	3	3850	69	-86.5
0	2	3900	63	-80.5
1	1	3950	0	-17.5
2	0	4000	44	-61.5
3	-1	4050	12	-29.5
4	-2	4100	59	-76.5
5	-3	4150	47	-64.5
1	2	5900	55	-72.5
2	1	5950	33	-50.5
3	0	6000	46	-63.5
4	-1	6050	35	-52.5
5	-2	6100	59	-76.5
0	4	7800	77	-94.5
1	3	7850	69	-86.5*
2	2	7900	56	-73.5
3	1	7950	12	-29.5
4	0	8000	57	-74.5
5	-1	8050	28	-45.5

To verify these results using the proper graphs:

- 1 Add a rectangular graph named “**Spectrum**”.
- 2 Add a power spectrum measurement using the settings in the following figure, then click **Apply**.



- 3 Run the simulation. The simulation response in the following graph should display after changing your graph's trace styles and axes to match those shown.



- 4 Right-click on the graph and choose **Marker** to measure the values in the graph and compare them with those given in the previous table of expected values. Move the "+" cursor to the point you want to measure and click to display the graph value at that point. The result is as expected, otherwise TST2IN would generate an error message. Alter any elements in either of the two text data files and run the simulation; an error message is generated.
- 5 Save and close the project





# INDEX

## A

ACPR 13-29  
Adding  
    a chip cap cell 5-18  
    measurements 8-15  
    ports and wires 3-10  
    subcircuits to diagrams 3-10  
    subcircuits to schematics 3-9  
AM modulation 8-7  
Animation 7-19  
Artwork cell  
    adding ports to 5-16  
    adding to a schematic element 5-10  
    assigning 5-10  
    creating 5-12  
AWR Design Environment 3-2  
    components 3-3

## B

Back annotation 6-8  
BER simulation 9-9  
Bias circuit 6-6  
Blocks 8-8  
    connecting 8-10, 8-11  
    placing in a diagram 8-9, 8-10, 9-2

## C

Cell libraries 3-15  
Circuit  
    analyzing 4-9  
    optimizing 4-14  
    tuning 4-10  
Component Browser 6-22  
Conductors, adding to layout 7-6  
Connecting nodes 3-10  
Creating

    layout 3-13  
    variables 9-9  
Creating RF Chain 12-1  
Currents, animating 7-18  
Curve meter 6-4

## D

Data file, changing the ground node 5-6  
Data types 8-1  
Database units 5-3  
De-embedding lines 7-11  
Default  
    grid size 5-3  
    project units 4-2  
Demodulation 13-7  
Distributed interdigital filter 7-2  
Documentation 1-3  
Dynamic load line measurement 6-21

## E

E-fields, viewing 7-18  
Electromagnetic (EM) simulator 7-1  
Element Browser 3-4  
Element catalog 8-10  
Element symbol, changing 5-6  
Elements, adding to schematics 3-8, 4-3  
EM simulation 7-1  
EM structure drawings 3-12  
EM structures  
    adding to schematic 7-28  
    creating 3-11, 7-2  
End-to-end 64QAM 13-20  
End-to-end system 9-1  
Exporting layout 5-26

**F**

Filters 10-2, 10-10  
     layout 7-20  
 FLEXlm license, obtaining 2-4  
 Frequency  
     center 8-4  
     sampling 8-4  
     simulation 4-7  
 FSK 13-1  
     modulator 13-4

**G**

GDSII cell library 5-4  
 Graphs  
     adding measurements 3-16  
     creating 3-16, 8-14  
     displaying results on 7-16  
     types 3-17  
 Ground node  
     adding 4-6  
     changing 5-6

**H**

Hardware/software requirements 2-2  
 Harmonic balance 6-1, 11-1  
     port 6-10, 6-11  
     simulator 11-1

**I**

I/Q imbalance 13-13, 13-14  
 Image rejection ratio 13-14  
 Importing  
     cell library 5-4  
     data file 5-5  
     GDSII cell library 5-4  
     layer process file 5-3  
     models 11-1  
 Installing MWO/VSS 2-1

**L**

Large signal reflection coefficient 6-13  
 Layer process file (LPF), importing 3-15, 5-3  
 Layout

    creating 3-13  
     creating from schematic 5-1  
     exporting 5-26  
     viewing 5-11

Layout cell, anchoring 5-11

Layout Manager 3-15

Linear simulators 4-1

Lines

    de-embedding 7-11

Lines, de-embedding 7-11

Load line measurement 6-21

Lumped element filter, creating 4-1

**M**

Measurements

    adding 3-17, 4-8, 8-15

    IP3 vs voltage 6-27

Microstrip elements, placing 5-7

Mixer modeling 13-32

MTRACE2 element 5-20

Multi-tone analysis 6-1

MWO/VSS design flow 3-1

**N**

Netlists, creating 3-5

Nonlinear

    measurements 6-1

    model 6-3

    simulation frequencies 6-12

    simulator 6-1

**O**

Online help 1-4, 3-19

Optimization goals, adding 4-14

Optimizing

    circuits 4-14

    simulations 3-18

**P**

Parameter

    editing 4-6, 8-13

    propagation 8-6

- tuning 9-8
- Ports
  - adding 3-10, 7-11
  - adding to artwork cell 5-16
  - and wires 3-9
  - editing 3-11
- Pout vs. frequency measurement 6-20
- Power amplifier circuit 6-2
- Project
  - creating 3-5, 4-1, 4-2, 8-7
  - opening 3-5
  - saving 3-5
  - setting default units 4-2
- Project Browser 3-4

## Q

- QAM 13-21
  - diagram 9-2
  - system simulation 10-7

## R

- Receiver 13-4, 13-7
- RF Budget Analysis 12-1
- RF Chain, creating 12-1
- Routing elements 5-20

## S

- Sampling frequency 8-2
- Schematic
  - adding elements to 4-3
  - copying 6-24
  - creating 4-3
  - editing 5-18
  - importing 6-15
- Scripts 3-19
- Signal representation 8-2
- Simulation
  - frequency 3-17, 4-7, 7-13
  - optimizing 3-18
  - results 8-17
  - tuning 3-18
- Single-tone analysis 6-1

- Snapping layout cells 5-21
- Starting MWO/VSS 3-2
- Subcircuits
  - adding to a diagram 3-10, 10-1
  - adding to a schematic 3-5, 3-9, 6-16
- Swept variables 6-27
- Symbol, changing 5-6
- System diagram 3-7, 8-10
  - adding blocks 8-10
  - creating 3-7, 8-9
  - linking to MWO 10-1
- System settings, defaults 8-8
- System simulator options 8-14

## T

- Technical support 1-4
- Test points, adding 8-11
- Third-order intermodulation measurement 6-26
- Tuning
  - a circuit 4-10
  - simulations 3-18
  - system parameters 9-8
- Two-tone harmonic balance port 6-24

## V

- Variables
  - creating 4-12, 9-9
  - sweeping 6-27
- Vector signal analyzer block 11-4
- Vector variables 4-15
- Vias, adding 7-9
- Viewing
  - a layout 5-11
  - E-Fields 7-18
  - structures in 3D 7-11
- VSS theory 8-1

## W

- Website support 1-4
- Wires, adding 4-4
- Wizards 3-19

